IN-64
104052
P-50

NASA Technical Memorandum 105612
ICOMP–92–01

Institute for Computational Mechanics in Propulsion (ICOMP)

6th Annual Report–1991

(NASA-TM-105612) INSTITUTE FOR COMPUTATIONAL MECHANICS IN PROPULSION (ICOMP) Annual Report No. 6, 1991 (NASA) 50 p

N92-27196

Unclas
G3/64 0104052

May 1992

NASA

INSTITUTE FOR COMPUTATIONAL MECHANICS IN PROPULSION
NASA Lewis Research Center
ICOMP
Case Western Reserve University
Ohio Aerospace Institute

NASA Technical Memorandum 105612
ICOMP-92-01

Institute for Computational Mechanics in Propulsion (ICOMP)

Sixth Annual Report—1991

Compiled and edited by
Dr. Charles E. Feiler
ICOMP Executive Officer

Approved by
Dr. Louis A. Povinelli
ICOMP Director

May 1992

CONTENTS

INTRODUCTION 1

THE ICOMP STAFF OF VISITING RESEARCHERS 3

RESEARCH IN PROGRESS 4

REPORTS AND ABSTRACTS 25

SEMINARS 33

WORKSHOP ON ENGINEERING TURBULENCE MODELING 40

PRECEDING PAGE BLANK NOT FILMED

INTENTIONALLY BLANK

INSTITUTE FOR COMPUTATIONAL MECHANICS

IN PROPULSION (ICOMP)

SIXTH ANNUAL REPORT

1991

SUMMARY

The Institute for Computational Mechanics in Propulsion (ICOMP) is a combined activity of Case Western Reserve University, Ohio Aerospace Institute (OAI) and the NASA Lewis Research Center in Cleveland, Ohio. The purpose of ICOMP is to develop techniques to improve problem-solving capabilities in all aspects of computational mechanics related to propulsion. This report describes the activities at ICOMP during 1991.

INTRODUCTION

The Institute for Computational Mechanics in Propulsion (ICOMP) was established at the NASA Lewis Research Center in September 1985, to improve problem-solving capabilities in all aspects of computational mechanics relating to propulsion. ICOMP provides a means for researchers with experience and expertise to spend time in residence at Lewis performing research to improve computational capability in the many broad and interacting disciplines of interest in aerospace propulsion.

The scope of the ICOMP program is: to advance the understanding of aerospace propulsion physical phenomena; to improve computer simulation of aerospace propulsion components; and to focus interdisciplinary computational research efforts. The specific areas of interest in computational research include: fluid mechanics for internal flows; structural mechanics and dynamics; and fluid-structural interactions. In 1990 a new focus was added with the formation within ICOMP of the Center for Modeling Turbulence and Transition (CMOTT). The organization and operation of ICOMP as originally constituted were described in ICOMP Report No. 87-8 (NASA TM-100225), "The Institute for Computational Mechanics in Propulsion, (ICOMP), First Annual Report," Nov. 1987. Since its inception, ICOMP employment contracts and payroll operations have been carried out by Case Western Reserve University. With the formation of the Ohio Aerospace Institute (OAI), several of ICOMP's program administrative functions including contracting, payroll, visa applications, and benefit administration have been transferred to OAI. Other aspects of ICOMP and its day-to-day operations are not changed by the addition of OAI to ICOMP. A brief description of OAI and its relation to NASA LeRC follow.

The Ohio Aerospace Institute is a not-for-profit corporation organized as a consortium of nine Ohio universities (University of Akron, Case Western Reserve University, University of Cincinnati, Cleveland State University, University of Dayton, Ohio State University, Ohio University, University of Toledo, and Wright State University), two federal laboratories located in Ohio (NASA Lewis Research Center, and Wright Laboratory) and an assortment of companies. OAI is dedicated to blending the capabilities of these three sectors together in a variety of collaborative initiatives. These involve aerospace related research, graduate education, and technology transfer.

In order to provide for the daily operation of the Institute, an ICOMP Director, Dr. Louis A. Povinelli, and an Executive Officer, Dr. Charles E. Feiler, have been officially appointed. Management of ICOMP by these two principles is supplemented by a Steering Committee. In addition, a member of OAI has been added to the Steering Committee. The current Steering Committee membership is: Professor E. Reshotko, Chairman; Dr. H. Brandhorst; Dr. L. Diehl; Dr. M. Goldstein; Professor I. Greber; Dr. T. Keith; Professor R. Mullen; Dr. L. Nichols; Dr. L. Pinson; and Dr. L. Reid.

The remainder of this report summarizes the activities at ICOMP during 1991. It lists the visiting researchers, their affiliations and time of visit followed by reports of RESEARCH IN PROGRESS, REPORTS AND ABSTRACTS published and SEMINARS presented. An overview summary of a special day-and-a-half long workshop on turbulence modeling, organized and conducted by CMOTT, is also presented.

THE ICOMP STAFF OF VISITING RESEARCHERS

The composition of the ICOMP staff during 1991 is shown in figure 1. Forty-nine researchers were in residence at Lewis for periods varying from a few days to a year. Figure 2 is a photograph of the ICOMP Steering Committee and the visiting researchers taken at a reception in July 1991. Figure 3 lists the universities or other institutions represented and the number of people from each. The figure lists thirty-eight organizations. Figure 4 shows the growth of ICOMP during its first six years in terms of staff size, organizations represented and technical output as measured by the numbers of seminars, reports and workshops. The next sections will describe the technical activities of the visiting researchers starting with reports of RESEARCH IN PROGRESS, followed by REPORTS AND ABSTRACTS, and finally, SEMINARS.

RESEARCH IN PROGRESS

Dare Afolabi, Purdue University, Indianapolis

The objective of my research at ICOMP was to gain insight into the mechanics of flutter, and other forms by which components of propulsion systems lose elastic stability. Using results from relatively recent developments in dynamical systems--namely, singularity theory, catastrophe theory, and bifurcation theory--it was found that eigenvalue degeneracy is a major cause of flutter. This arises when two eigenvalues of opposite Krein signatures collide. Not all eigenvalue degeneracies give rise to flutter. For example, when eigenvalues of the same signature meet, they simply pass through each other. In such cases, they lead to loss of geometric stability, rather than elastic stability. The above findings as applied to propulsion hardware were presented at an ICOMP seminar entitled, "Elastic Stability of Rotating Systems and Catastrophe Theory," given on July 1 and contained in the ICOMP Report entitled, "Modal Interaction in Linear Dynamic Systems Near Degenerate Modes."

Suresh K. Aggarwal, University of Illinois, Chicago

The computation of confined laminar gaseous and spray flames were continued. In addition, the efforts were initiated for the computation of strongly swirling confined turbulent gaseous and spray flows. The first part of the study deals with the numerical simulation of Burke-Schumann type flames with gaseous as well as with liquid spray fuels. The simulations not only show excellent agreement with the analytical Burke-Schumann analysis, but also highlight the effects of the assumptions made in the analysis. Details are provided in the publications listed. The efforts for the computation of strongly swirling turbulent flows had a limited success and are still continuing in order to predict the recirculation zone and other features of the strongly swirling turbulent jets. Results of this effort will be discussed in a future ICOMP publication. A significant part of the time was spent in consultation with Dr. M. Mawid, who was working as a Research Associate at ICOMP, and with Dr. E. J. Mularz and Dr. D. L. Bulzan of NASA Lewis Research Center.

PUBLICATIONS

Aggarwal, S. K.; and Mawid, M.: A Detailed Numerical Investigation of Burke-Schumann Gaseous and Spray Flames. AIAA Paper 91-2311, 1991.

Aggarwal,S. K.; Bulzan, D. L.;and Mawid, M.: On the Structure of Gaseous Confined Laminar Diffusion Flames/Numerical Investigation. To be published in Combust. Sci. Technol., 1992.

Aggarwal, S. K.; Bulzan, D. L.; and Mawid, M.: On the Predictions of Burke-Schumann Type Spray Flames Supported by Droplet Streams. To be published in Phys. Fluids, 1992.

Kyang Ahn, (Postdoctoral) Cleveland State University

I worked on numerical simulation of the 2D turbulent oscillating flow in Stirling engine heat exchangers. The research includes estimating the effect of imposed unsteadiness on the time-mean turbulent structures using a standard κ-ε model and also determining the criterion for which quasi-steady turbulence modeling is valid. Computationally, a 2D finite-volume code was modified to handle oscillating flow and was used to simulate various design operating conditions of the Stirling engine. Comparisons of the numerical results were made with the available experimental data. After showing reasonable agreement with the experimental data, the quasi-steady criteria was determined purely numerically by simulating more than 25 different operating conditions; it was concluded that most of the operating conditions of NASA's space power Stirling engines are in the non-quasi-steady regime while the operating conditions of the engine developed by other commercial firms are in a quasi-steady regime.

Fred A. Akl, Louisiana Tech University

Occam implementation of an 8-node quadrilateral plane strain/plane stress element has been accomplished. The new routine has been added to the binary tree eigensolver previously developed on the

transputer network to solve for the subspace of the lowest order eigenpairs of the generalized eigenproblem $[K]\ [\Phi] = [M]\ [\Phi]\ [\Lambda]$. In addition, a number of changes have been made to allow the solver to utilize, if necessary, a relatively smaller size binary tree configuration of processors on a given size of network. One current limitation of the solver which it may be desirable to eliminate in the future is the requirement that the number of finite elements must be equal to 2^n, where n = the depth of the binary tree.

Thomas F. Balsa, University of Arizona

The central theme of this research is an analysis/numerical computation of the nonlinear development of a low-frequency instability mode in a supersonic mixing layer. The analysis is based on the ideas of Goldstein on nonequilibrium critical layers. Using these ideas we simplify the Navier-Stokes equations to a system of coupled (two) PDE's and (one) ODE. These equations describe, respectively, the evolution of the vorticity and temperature fields and the amplitude of the instability mode. The nonlinear coupling between these field equations arises because of the generation of vorticity by the baroclinic torque in a compressible fluid.

Our principal work this summer concentrated on the development and coding of a numerical scheme for the solution of this problem. The numerical scheme is second order accurate in space and time; it is a combination of a predictor-corrector method for the u-convection term and a Crank-Nicholson method for the viscous terms. The v-convection terms are treated implicitly together with the viscous terms. The method gives very good results at moderate Reynolds numbers; for inviscid flows a Lagrangian technique was also refined.

The main result of this work shows that the mixing layer will roll-up at supersonic speeds into distinct cores of vorticity. The braids that link these cores contain large amounts of vorticity generated by the baroclinic torque. As the Mach number is increased (say to about 5) the vorticity in the braids is confined to narrow and elongated islands which will be very unstable to small scale disturbances. Under these conditions it is expected that the mixing layer will quickly become turbulent.

Tawit Chitsomboon, (Postdoctoral) Old Dominion University

A 3D k-ε turbulence model implementation to RPLUS3D was completed. A test run on a 3D corner flow was successful. The case of a supersonic jet in a supersonic cross flow, however, was not successful. The failure was attributed to the fact that the turbulence model was basically designed for attached boundary layer flow. Many modifications to the basic scheme were tried but none was found to fix the problem. A new code named MAWLUS (Multigrid Acceleration With L-U Scheme) has been spawned out of RPLUS3D. The code is almost fully written. Among other new features are: code lines reduced from about 10,000 to 6,000; multigrid capability; grid sequencing capability; multiblock capability where each block has its own index system independent of the other blocks; option for implicit boundary condition; and option for 2D flow. A test run on a laminar, supersonic flow over a 10 degree ramp gave accurate results. The shock front was captured to within one grid point without noticeable oscillations. This is rather surprising since the code is still based on a central differencing scheme. The results obtained by a multigrid procedure were equally accurate but did not show a faster convergence rate. More scrutiny into the fine detail of the multigrid procedure is obviously needed.

As of October 1, 1991, the investigator was switched from the National Aerospace Plane project into the High Speed Civil Transport project, in conjunction with the Nozzle Technology Branch, Propulsion System Division. This new project will involve applying the MAWLUS code to solve for the flowfield within and surrounding the exhaust-ejector system of an HSCT engine. The initial effort has been spent in modifying and testing MAWLUS to suit the particular needs of the said flowfield. In particular, subsonic inflow and outflow boundary conditions were implemented and tested; 3D multiblock procedure was also tested. The first real run was set up with the 127x35x100 grid given to the investigator by Dr. Yunho Choi of Sverdrup Technology. For convenience in setting up boundary conditions, the single grid was divided into three blocks. Currently, the run fails at a very early stage. Efforts are being made to achieve a successful run.

The plans for the year 1992 are: use the k-e turbulence model to solve the turbulent flowfield and compare the solution with that obtained by using an algebraic turbulence model as well as a laminar solution; conduct parametric studies as needed, like changing the total pressure ratio; and improve the multigrid acceleration procedure.

Frederic Coquel, ONERA

An increasing effort is being devoted to the design of numerical Flux Vector Splitting (FVS) methods well suited to the approximation of the Navier-Stokes equations. As is well known, classical FVS methods, such as the celebrated Van Leer splitting, fail in preserving contact and shear discontinuities. This drawback prevents these methods from giving accurate boundary layer representation in viscous calculations. Current research therefore, addresses the problem of deriving low diffusive FVS-like methods while retaining the robustness and computational efficiency of the Van Leer scheme.

A very promising trend shown by the work of Meng-Sing Liou, consists of mixing Flux Difference Splitting (FDS) and FVS approaches. In a joint work, we investigate the stability and "monotonicity" of several hybridizations that exactly preserve steady contact discontinuities. The results obtained so far strongly support the interest in mixed FDS and FVS schemes. This research is carried out with particular attention paid to the extension for multi-component and multi-temperature problems. Indeed, we focus on algorithms that preserve, by construction, positivity of mass fractions and vibrational energies

Stephen Cowley, University of Cambridge, England

During this visit to ICOMP I continued working with Marvin Goldstein and Stewart Leib on a problem concerning vortex stretching in steady flow past the leading edge of a thin blunt body. Upstream it is assumed that the uniform flow is slightly disturbed in such a way that steady vortex lines are induced normal to the thin body. These vortex lines get snared by the front of the plate, with the result that vortex stretching occurs. Near the leading edge of the body, viscosity is assumed to prevent the vortex stretching from becoming nonlinear. However, sufficiently far downstream the induced spanwise velocity is governed by the nonlinear, kinematic-wave equation. As a result a singularity develops, which we interpret as inducing bursting from the boundary layer. An analytic description of this bursting was derived -- a possible interpretation is that it is an example of open separation. Most of my visit was spent putting the final touches to a paper, which has now been accepted by the Journal of Fluid Mechanics.

D. Davis, (Postdoctoral) University College London, England

I am considering the growth of 3D Rayleigh disturbances in an incompressible (primarily Blasius) boundary layer over a flat plate possessing a small adverse pressure gradient. There is early analytical evidence that, above a certain frequency-dependent value of the scaled adverse pressure gradient the maximum growing linear mode does not occur as a plane wave (as is generally believed) but instead as a 3D wave whose obliqueness is determined by the size of the pressure gradient.

Comparisons with computational solutions of the Rayleigh equation for 0(1) adverse-pressure-gradient boundary layers are currently underway. The main codes were supplied by Dr. Lennart Hultgren and Dr. Stewart Leib.

Robert J. Deissler, Los Alamos National Laboratory

There are two projects, Taylor-Couette Flow and Rayleigh-Benard Convection with Through-Flow, which have dominated much of my attention during the past year. In the Taylor-Couette system, a fluid exists between two concentric cylinders where the inner cylinder is rotating and where there is a through-flow parallel to the axes of the cylinders. For sufficiently large rotation rate the steady background flow is unstable and rolls (or Taylor vortices) form. Depending on the axial flow rate the instability can be either *absolute* (meaning that a localized perturbation grows at a given point in the laboratory frame) or, if the axial flow rate is sufficiently large, *convective* (meaning that the perturbation grows only in a moving frame of reference, i.e. a frame of reference moving in the axial direction, eventually damping at any given point in the laboratory frame). Under convectively unstable conditions noise near the entrance region will be amplified resulting in spatially growing waves and a *noise-sustained structure* [Deissler, R. J., J. Stat. Phys. **40** (1985) 371; Physica D **25** (1987) 233; J. Stat. Phys. **54** (1989) 1459]. This is an ideal system to study the effects of noise and noise-sustained structure since a convective instability can exist for fairly low flow rates, meaning that the system is easily controlled and that there is no turbulence. This, of course, makes the solution of the Navier-Stokes equations much easier. Another advantage is that the Ginzburg-Landau equation can be derived for this system,

meaning that direct comparison between the Ginzburg-Landau system and Navier-Stokes system can be made.

Dr. Wai-Ming To and myself have been working on a code to solve the Navier-Stokes equations for this system. Just recently we have gotten one code running which uses Chebychev polynomials in the radial direction and Fourier transforms in the axial direction. Here we are assuming no azimuthal dependence for the velocities, which is correct for through-flow velocities that are not too large. The diffusive part is solved implicitly (for stability) and divergence of the velocity is imposed to zero at $t + \Delta t$.

Presently, Taylor-Couette flow with through-flow is being experimentally studied at the University of California at Santa Barbara by Ken Babcock, et al. They have submitted a paper for publication in which they have found noise-sustained structure [Ahler, G.; Babcock, K. L.; and Cannell, D.: "Noise-Sustained Structure in Taylor-Couette Flow with Through-Flow," 1991]. Our code has successfully reproduced some of the experimental results such as the boundary between convective and absolute instability. However, in order to better study noise-sustained structure, it is necessary that we use finite differencing in the axial direction. The code is now being modified to this effort.

For higher through-flow rates, spiral rolls occur. Our code will be able to be generalized to include azimuthal dependence by taking Fourier transforms in the azimuthal direction. We will be able to study this phenomena, and again be able to make direct comparison with experiment.

The other system for which Wai-Ming To and I are writing a code is Rayleigh-Benard convection with through-flow. This system consists of a horizontal rectangular pipe which is heated from below and through which fluid flows. This system shares many of the properties of the Taylor-Couette system. For a sufficiently large vertical temperature gradient, rolls form. Depending on the through-flow rate the system will be either absolutely or convectively unstable, and the system can be very well controlled since turbulence does not exist for the Reynolds-numbers of interest. Also, under convectively unstable conditions, noise-sustained structure can exist. This system also exhibits behavior with no analogue to behavior in the Taylor-Couette system. Namely the simultaneous existence of both transverse and longtudinal rolls in different parts of the cell for a range of parameter values. A difficulty in writing a numerical code to solve the Navier-Stokes equations for this system is that there are solid wall boundaries in two directions. This is one reason we are now concentrating on the Taylor-Couette problem. However, we have recently had some insights which should make a 3D code for this system workable. Once we get the code running we can make direct comparison with the experiments being done at the University of California at Santa-Barbara by Steve Trainoff, et al. Also, we can calculate coefficients for a system of coupled Ginzburg-Landau equations which describe this system, the transverse rolls being represented by one equation and the longitudinal rolls being represented by the other equation [Ahlers, G.; Brand, H. R.; and Deissler, R. J.; Phys. A **43** (1991) 4262].

I have started another project on Stable Localized Solutions in a Thin Film on the Underside of a Cooled Horizontal Plate with Dr. Alexander Oron from Technion in Israel, during his visit here. We have been studying a 2D equation which describes the fluid-air interface of a thin liquid film on the underside of a horizontal plate, which is cooler than the ambient air. Because of the cooling, the Marangoni effect - which is very important in low gravity conditions and therefore in processes of space technology - is stabilizing and we have found that stable axisymmetric localized solutions exist. It will be very interesting to see whether such stalactite-like structures exist in actual experiments. A paper has been submitted for publication.

The Eckhaus and Benjamin-Feir Instability Near a Weakly Inverted Bifurcation is a study performed in collaboration with Dr. Helmut Brand while at Los Alamos. The project was completed during my stay here at ICOMP and a paper has been submitted for publication. The basic result is that the conditions for instability of a spatially periodic or spatially and/or time periodic state (i.e. secondary instability) were derived for systems near a subcritical bifurcation. This is important since the bifurcation in many systems is subcritical, an example being binary fluid convection, i.e. two miscible fluids such as water and alcohol with a vertical temperature gradient.

A. O. Demuren, Old Dominion University

Current research work focuses on implementation of second-moment turbulence closure models suitable for application to complex 3D flows. The most promising are the full Reynolds Stress Models (RSM) based on the numerical solution of partial differential equations for the turbulent (Reynolds) stresses and associated turbulence fluxes, and a representative equation for the length scale. In earlier work, the model proposed by Launder, Reece, and Rodi (JFM, 1975) was generalized for curvilinear coordinates and incorporated in an

incompressible multigrid code to ensure efficient convergence on any grid size. Demuren (1990a, 1990b) and Claus and Demuren (1991) have shown that the full Reynolds stress equations can be solved with little difficulty and minimal additional cost if the stiffness of the system of equations is reduced by splitting of the source terms into implicit and explicit parts. The resulting system is more robust and less sensitive to model coefficients. More sophisticated Reynolds stress models which satisfy to a greater degree required kinematic constraints and thus offer promise of better accuracy are now being explored. These second-moment closure models will also be extended for application to compressible flows through the incorporation of models for additional terms which arise such as pressure-dilatation, density-velocity, density-temperature correlations, etc. The models are to be incorporated in existing LU and ADI codes for both 2D and 3D flows. Incorporation into existing codes has been delayed because current versions were found to converge too slowly when the k-ε model was implemented in them. It is now necessary to introduce convergence acceleration techniques into these codes before much progress can be made with the turbulence modeling.

REFERENCES

Demuren, A. O.: Calculation of 3D Turbulent Jets in Crossflow with a Multigrid Method and a Second-Moment Closure Model. NASA TM-103159, 1990.

Demuren, A. O.: Characteristics of 3D Turbulent Jets in Crossflow. Symposium on Turbulence, 12th, Rolla, MO, 1990, X.B. Reed, Jr., G.K. Patterson, and T.L. Zakin, eds., University of Missouri-Rolla, 1990, pp. B1-1-B1-11.

Claus, R. W.; and Demuren, A. O.: Numerical Calculations of Jets in Crossflow: Effects of Numerical Accuracy and Turbulence Model. In preparation.

J. S. B. Gajjar, University of Exeter, England

The work that I did this year was a continuation of that initiated last year on the nonlinear evolution of an oblique mode propagating in a compressible boundary layer. An asymptotic analysis was used based on the assumptions of long waves, small phase-speeds and small growth rates, to derive the nonlinear unsteady critical layer equations describing the evolution of the mode for both insulated and noninsulated boundary layers. The assumptions made restrict the analysis to describing the evolution of the first mode of instability and thus for low Mach numbers. Various comparisons were made of the predicted growth rate with numerical solutions of the compressible Rayleigh equation with a code provided by Dr. L. Hultgren. The comparisons were reasonably accurate for Mach numbers up to 1.5. The nonlinear equations were solved and several results obtained. Part of this work is to appear in the proceeding of the Royal Aeronautical Society meeting on "Boundary Layer Transition and Control" in Cambridge in April 1991. A fuller version of the paper is currently being written up for submission to a journal.

Max D. Gunzburger, Virginia Tech.

We have developed some potentially good algorithms for flow control and optimization; numerous computational experiments have been carried out. We are in the process of furthering the development of these methods and of applying them to flow control and optimization problems that are of relevance to the Lewis mission. We have investigated two classes of methods. First, we have devised algorithms based on co-state methods. Second, we have looked at methods based on the direct calculation of sensitivities. In the past, we have been able to give many of these methods a firm analytical foundation, and in many cases as well, we have implemented the methods into computer codes. Among the optimization and control objectives we have studied are drag minimization, avoidance of high-temperature hot spots, flow tracking, and enhancing or deterring mixing. Among the control mechanisms we have studied are blowing/suction, heating/cooling, and domain shape. Many other objectives and controls can also be possibly treated by our techniques. One of the aims of our work at ICOMP is to extend our methods to other objectives and controls that would be of interest to NASA Lewis.

We are also embarking on a study of least-squares finite element methods for a variety of problems involving linear and nonlinear partial differential equations. Such algorithms have been studied and developed at ICOMP by B. Jiang and C. Chang. Their work has produced some new algorithms that are superior to any

existing methods for some of the classes of problems that they have treated. Our own contribution will be to analyze and further develop these algorithms for nonlinear problems.

L. Hakan Gustavsson, Lulea University of Technology, Sweden

The main part of the research has been devoted to a study of the development of localized disturbances in parallel shear flows and particularly the conditions under which it is possible to counter algebraic growth by prescribing a certain amount of normal vorticity in the initial disturbance.

The growth of normal vorticity of a localized disturbance has previously[1] been found to lead to a substantial amplification of the kinetic energy of a perturbation, a mechanism thought to be involved in bypass transition processes. Therefore, it is of great interest to know whether the growth can be countered by a proper amount of initial vorticity. The study has been concentrated to scales highly elongated in the streamwise direction for which the largest amplifications have been found. The results of prescribing the initial vorticity show that the energy growth can be reduced by as much as 40%. This result may be useful in contexts where three-dimensional disturbances appear naturally in a flow and it is of interest to minimize their effects on momentum and heat transfer.

REFERENCE

Gustavsson, L. H.: Energy Growth of 3D Disturbances in Plane Poiseuille Flow. J. Fluid Mech., vol. 224, Mar. 1991, pp. 241-260.

Thomas Hagstrom, University of New Mexico

In my stay at ICOMP, I carried out research on a variety of issues in computational fluid dynamics. With John Goodrich of NASA, I worked on completing numerical experiments on outflow boundary conditions for incompressible flows. We are working on four benchmark problems which have been proposed. The boundary condition construction is based in part on the work I have done here during past visits. Partial results are to be presented at a workshop at Stanford this July, while the complete results will be reported at a later date. With S. I. Hariharan of ICOMP, I continued my work on radiation boundary conditions for time accurate simulations of compressible flows. In particular, we completed a theoretical study of improved conditions for the wave equation, and began the work of translating them to the Euler equations and incorporating them into our existing code. We also found a number of other ICOMP researchers who may be interested in trying them out. Thirdly, with John Goodrich (NASA) and Krishnan Radhakrishnan (Sverdrup), plans were made to begin the development of a large scale code for solving low Mach number reacting flow problems. We plan to have a preliminary version ready soon, which will solve reaction-diffusion equations for a number of reacting species and temperature, including thermal diffusion and other physical effects. This will later be coupled with a low Mach number flow code. We hope to study flame instabilities and other dynamic combustion phenomena.

S. I. Hariharan, University of Akron

During this period several theoretical as well as computational questions governing time accurate calculations of exterior hyperbolic problems in two dimensions were addressed. The year began with a question of if it is at all feasible to obtain time accurate computational procedures for hyperbolic problems in exterior regions. Specifically this question is pertinent only to 2D problems. This question is centered around time accurate calculations of unsteady flow past airfoils. Several model problems were considered to analyze this phenomena. It was realized that the computational analysis of 2D hyperbolic problems should include a component that insures time accuracy. This component is related to the decay properties of the 2D problems. In contrast the corresponding 3D problems do not require these considerations. Analysis of these difficulties motivated us to look into fundamental problems governed by the wave equation in the frequency domain as well as in the time domain. The analysis included designs of appropriate absorbing boundary conditions to maintain time and frequency accuracies. These resulted in conditions that contain coefficients as functions of time for the time dependent wave equation. Extensions to Euler equations were made and the computational verifications are in progress.

M. E. Hayder, (Postdoctoral) Princeton University

The goal of my present research at ICOMP is to compute aerodynamic noise in a supersonic jet. A numerical model based on the Navier-Stokes equations is under development for these simulations. In developing the numerical model, it is assumed that large scale structures in the flow field dominate the sound production. As a consequence of this assumption, we seek large eddy simulations with a subgrid turbulence model, instead of full DNS computations.

Different numerical schemes are being explored for an efficient and time accurate code. At present, a model for a plane jet which is second order in time and fourth order in space accuracy is under testing. This model is also being extended for three dimensional round jet computations. Spectral discretization in the azimuthal direction is planned for the round jet simulations. Other candidate schemes for our model are compact and ENO (essentially non-oscillatory) schemes.

Joe Iannelli, University of Tennessee

During my July visit at ICOMP two related topics were assiduously pursued: (1) an innovative analytical formulation for a neutral five-species equilibrium-air equation of state; and (2) a thorough analysis of average state determinations for general equilibrium-air and curvilinear-coordinate Euler fluxes.

The developments for the former topic identify an efficient two-equation system which leads to the five equilibrium mass fractions, temperature, pressure, and their respective partial derivatives. A detailed technical report has been prepared and reviewed by a NASA Lewis resident scientist, and the final revised version has just been completed.

Concerning the second topic, modern split-component evaluations of the flux vector Jacobians are thoroughly analyzed for equilibrium-gas average-state determinations. It is shown that all such determinations satisfy a fundamental eigenvalue consistency theorem. A conservative-variable average state which admits an unique classical average density is then developed for arbitrary equilibrium-gas equations of state and curvilinear coordinate fluxes.

Original expressions for eigenvalues, sound speed, Mach number, and eigenvectors are then developed in terms of the pressure Jacobian. These relations show that the form of the average eigenvalues, Mach number, and eigenvectors may not coincide with the classical pointwise forms.

A general equilibrium-gas equation of state is then discussed for conservative-variable CFD Euler formulations, and unique compatibility relations are identified that constrain the pressure Jacobian derivatives. Thereafter, alternative forms for the pressure variation and average sound speed are developed in terms of two average pressure Jacobian derivatives. Significantly, it is shown that no additional degree of freedom exists in the determination of these two average partial derivatives of pressure. On the contrary, these two derivatives are simultaneously computed exactly without any auxiliary relation, hence without any solution projection or arbitrary scale factors.

Several alternative formulations are then compared and key differences high-lighted with emphasis on the determination of the pressure variation and average sound speed. The relevant underlying assumptions are identified, including some subtle approximations that are inherently employed in published derivations.

Finally, a representative test case is discussed for which an intrinsically exact average state is determined. This exact state is then compared with the predictions of recent methods, and their inherent approximations are appropriately quantified.

All the analytical developments are documented in a revised comprehensive report which has also been completed recently.

J. Mark Janus, Mississippi State University

My two week visit at ICOMP/NASA Lewis involved several investigations, many of which have only touched the surface of the topic which they address. The primary focus of my visit was on the study of a model problem which "could" contain a relative-motion subdomain interface (i.e., an interior boundary condition). The problem is one which Dennis Huff has studied to demonstrate the need for accurate (nonreflecting) boundary conditions in turbomachinery flow simulations. It involves a cascade of flat plates undergoing very small amplitude oscillations creating acoustic disturbances throughout the domain. These disturbances propagate upstream and downstream and should pass "through" the inflow and outflow boundaries. Similarly, for the

domains of my concern, these acoustic waves should pass "through" a relative-motion interior boundary undisturbed. During my visit, initial software was developed for studying model problems of this type. The starting point was a code developed for the simulation of 3D rotating machinery. This code was modified for 2D cascades undergoing small amplitude oscillations. Presently, the code is capable of simulating an oscillating cascade with the downstream portion of the domain either fixed or in relative motion with the subdomain containing the flat plates. The cascade operates with zero interblade phase angle, creating planar disturbances moving upstream and downstream. It is anticipated that the code will also be capable of nonzero interblade phase angles with a few modifications. Although the results at the time of this report are inconclusive with regard to interior boundary efficacy, test runs have been completed which confirm the distortion of acoustic waves due to grid motion. This is not unexpected yet further study is warranted to determine the extent of the distortion.

In addition to the primary purpose for my visit, several other tasks were undertaken while at NASA Lewis. For example, another accomplishment was the identifying of the source of the improved performance of the "modified" LU 2-pass (plus-minus) approximate factorization to that of the standard 2-pass AF scheme. This was accomplished by examination of the matrix resulting from the product of the lower and upper operators. It was found that the modified 2-pass removed all AF error from the off-diagonal block elements, forcing it into the diagonal block elements. This led to the development of a scheme which removes even more AF error in an attempt to produce an "optimized approximate factorization" scheme. Convergence studies show improved performance of the new scheme although it presently does not seem to be a cost-effective improvement (too much additional effort involved in removing the additional error terms). This too will require further investigation.

An additional accomplishment from my visit was the modification of SSD I/O in the code developed by Dennis Huff for simulating unsteady cascades. A new routine was written to manage all SSD I/O using unblocked file structure. Significant reductions in system overheat cpu time was achieved. Also, discussions with NASA personnel were held to determine the potential of modifying the present 3D turbo flow code to incorporate the use of the linearized Euler equations in the solution strategy. Further study in this area is planned.

San-Mou Jeng, University of Tennessee Space Institute

The objective of this research is to develop a CFD-based model for bipropellant spray combustion in a small rocket engine. The computer code developed at the University of Tennessee Space Institute by Larosiliere and Jeng[1] has been extensively used to study small rocket combustion chamber performance for different liquid propellants (LOX/N_2H_4, N_2O_4/N_2H_4 and N_2O_4/MMH). The results of this research effort, including the transient behavior of combustors using different injector designs, has been summarized in a paper to be presented at the 1992 JANNAF Propulsion Meeting.

A significant portion of my research effort was to revise the existing CFD code described in Reference 1. It was intended to establish a new CFD code which is computationally efficient and robust, and which is able to calculate all-speed, two-phase and reactive flows. The traditional approach using Eulerian coordinates to describe gas-phase flows and Lagrangian coordinates to describe liquid-phase flows was considered.

Two major tasks were conducted to achieve the proposed research goal. The first task was to select a state-of-the-art CFD solver which can predict the time-dependent behavior of all-speed, single-phase, and reactive flows, and which is suitable to be upgraded for spray combustion calculations. A code[2] recently developed at the NASA Lewis Center was evaluated and selected. This code has been modified for rocket combustor calculations considering multi-fuel/oxidizer-inlets, complex rocket geometry and chemical kinetics of the considered propellants. This code has also been verified against a fuel-film-cooled H_2/O_2 rocket solver for spray combustion. This code has been fully debugged for non-evaporating droplets and has been fully incorporated into the gas-phase solver. Capabilities of the developed code have been demonstrated in predicting turbulent dispersions of different sized droplets in rocket combustion chambers.

REFERENCES

Larosiliere, M.; and Jeng, S. M.: "Bipropellant Spray Combustion Modeling in Small Rocket Engines," AIAA-91-2197.

Shuen, J.; and Chen, K.: "A Time-Accurate Algorithm for Chemical Non-Equilibrium Viscous Flows at All Speed," (to be published in J. Comp. Phys.)

Bo-Nan Jiang, University of Texas, Austin

It is well accepted that there is no unified computational method which can cover the whole range of flow problems. One must use different schemes for different categories of problems. For example, finite difference practitioners use artificial dissipation, upwinding or TVD-type schemes for capturing shocks in compressible flows, and a SIMPLE-type scheme to deal with incompressible viscous flows. In the finite element community, the classic Galerkin method is used for potential flows, the mixed Galerkin method and the Petrov-Galerkin method are developed for convective transport problems and compressible flows.

We have been developing a universal method for all seasons, namely the least-squares finite element method (LSFEM) for fluid dynamics problems.

The LSFEM of interest here is based on minimizing the residuals in first-order partial differential equations. Fortunately, the governing equations, which are derived from conservation laws and constitutive laws, are of first-order for most flow problems. For historic reasons (convenience for hand calculation and analysis), the equations in a first-order system are combined into a high-order partial differential equation (or equations) with fewer unknowns. We believe that in the computer age this transformation is unnecessary. The LSFEM can solve directly the original governing equations and give a reasonably good approximate solution for all unknown variables. It does not matter whether the partial differential equations are elliptic, parabolic or hyperbolic.

For convective transport (hyperbolic) problems, all existing conventional numerical methods will produce approximate solutions which either oscillate or smear out a sharp front. Finding accurate approximations for the discontinuous solutions of hyperbolic equations has been a persistent difficult task in computational fluid dynamics.

For pure convection problems, we developed an iteratively reweighed LSFEM which captures discontinuities in bands of elements that are only one element wide on both coarse and fine meshes. The solution of this method has neither smearing nor oscillation, and has superior accuracy.

The solution of the incompressible Navier-Stokes equations is more often undertaken using the velocity-pressure formulation. The Galerkin mixed finite element method based on the velocity-pressure formulation requires the satisfaction of the Ladyzhenskaya-Babu ka-Brezzi(LBB) condition which excludes the application of simple equal-order elements. Although for 2D problems quite a few convergent pairs of velocity and pressure elements have been developed, most of these combinations employ basis functions that are not convenient to implement. For 3D problems this difficulty becomes more severe and only rather elaborate constructions can pass the LBB test. For the same reason, finite difference methods must use staggered grids to avoid an oscillatory pressure solution. However, the use of a staggered grid with generalized coordinates is rather cumbersome. Another difficulty associated with the Galerkin mixed method is that the resulting linear algebraic equations are non-symmetric and non-positive-definite, therefore direct Gaussian elimination rather than iterative techniques has been considered the only viable method for solving the system. However, for 3D problems the computer resources required by a direct method become prohibitively large. The Petrov-Galerkin method also results in a non-symmetric matrix system, and relies on the choice of upwind parameters.

For the solution of incompressible viscous flow, the h-version of the LSFEM has been developed. This method is based on the velocity-pressure-vorticity formulation. The LSFEM leads to a minimization problem rather than to a saddle point problem, and the choice of combination of elements is thus not subject to the LBB condition. This method has an optimal rate of convergence for all variables. Since no derivatives are involved, the implementation of boundary conditions is extremely easy. In this method there is neither upwinding nor artificial dissipation nor arbitrary parameter. The resulting matrix is symmetric and positive-definite, therefore simple matrix-free iterative methods can be used for solving large-scale problems, and vectorization and parallelization are trivial. This year the 2D p-version of this method was developed (in collaboration with Dr. Sonnad of IBM Austin).

For the last two decades much effort has been devoted to developing mixed finite element methods which involve the simultaneous approximation of the principle unknown and its derivatives. Unfortunately, the accuracy of the derivatives is one-order lower than that of the primal variable in mixed methods. The usual least-squares method has the same problem. Also mixed methods have many other troubles mentioned above. Mixed methods and the usual least-squares method are based on the reduction of the original high-

order equation into several lower-order equations. We found that this simple procedure of reduction destroys ellipticity and thus the solution of mixed methods and the usual least-squares method cannot be optimal for all variables. Our theoretical analysis and numerical experiments show that by introducing the compatibility equation into the system of lower-order equations, we obtained an optimal LSFEM.

B. P. Leonard, University of Akron

In 1983 at the CTAC meeting in Sydney, Professor A. R. Mitchell of Dundee University coined the phrase "the ultimate embarrassment" referring to computational-fluid-dynamicists' inability to simulate the (apparently) simplest of flows: pure one-dimensional advection of a scalar profile in a constant velocity field. Most standard schemes (used in many research and commercial codes) fail miserably when applied to this test! First-order schemes are highly diffusive, second-order schemes terribly wiggly, and even the popular TVD schemes generate embarrassing steepening-and-clipping distortions. In the one-dimensional case these problems can be solved by using higher-order upwinding in combination with an universal limiter together with a discriminator to resolve narrow peaks (Computer Methods in Applied Mechanics and Engineering 88, 1991). The embarrassment is now focused on two (and three) dimensions. Even for uniform advection oblique to the grid, one sees anisotropic distortion and either gross diffusion (for first-order methods) or oscillation (higher order). Isotropy can be maintained by using a genuinely two-dimensional uniformly third-order polynomial interpolation algorithm (UTOPIA); but small undershoots (overshoots) remain. TVD schemes are monotonicity preserving but produce bizarre anisotropic distortion, in addition to serious clipping. A truly multi-dimensional limiter seems elusive, although some headway has been made. The steady-state case is in much better shape. This is shown in an ICOMP report "ULTRA-SHARP Solution of the Smith-Hutton Problem" (co-authored by Simin Mokhtari) which has also been submitted for journal publication. This paper also demonstrates the dismal performance of the popular "Hybrid" method of Spalding and Patankar's "power-law" differencing scheme. In particular, these methods (compute but) switch off the turbulence model wherever the component grid Reynolds number is larger than 2 (Hybrid) or about 6 (power-law) - which is almost everywhere, in practical cases!

W. W. Liou, (Postdoctoral) Pennsylvania State University

The goal of my research project is to develop second-order Reynolds stress closures for compressible turbulent flows. The progress made during the current reporting period is summarized in a NASA Technical Memorandum*. Using the Reynolds and Favre decompositions, Dr. T-H. Shih and I have extensively re-examined the complete, ensemble averaged mean equations and the equations for turbulent fluctuations and their second-order moments. A few advances in the modeling of compressibility effects were also studied. Recently, we have begun developing models to estimate the scaling associated with the different mechanisms in compressible turbulence. These models may be efficient for engineering calculations of compressible turbulent flows. They will also be used in constructing models for the transport of Reynolds stresses in compressible turbulent flows.

*Liou, W. W.; and Shih, T-H.: On the Basic Equations for the Second-Order Modeling of Compressible Turbulence. NASA TM-105277, 1990.

Joseph T.C. Liu, Brown University

During the June visit, a seminar was given on the subject of the Application of Coherent Structure Theory to Problems in Propulsion Systems. Interaction with Dr. R. R. Mankbadi was carried out on turbulence closure applied to turbulent shear flows subjected to imposed periodicities. The ideas of rapid distortion theory and coherent structure theory are synthesized in a simple closure involving the ratio of the phase-averaged kinetic energy to the shear stress, including discussions of a model equation for the phase-averaged viscous dissipation. Interactions across Divisional lines were also carried out, including discussions with NASA personnel on the use of coherent structure ideas towards enhancing mixing and decrease of NO_x in combustors, and in rapid mixing in ejectors/nozzles. This visit has lead to continued interactions with NASA Lewis.

Sherwin Maslowe, McGill University

My primary effort during a one and a half week visit to ICOMP was to initiate development of a theory to describe the propagation of wave packets in supersonic mixing layers. This theory differs from previous wave packet formulations in that wave packet effects are dominant in the critical layer rather than viscosity, time dependence, or nonlinearity. Such an approach was applied recently to homogeneous and stratified shear flows and will be described in a forthcoming paper with Professor D. J. Benney.

During my visit to ICOMP, it was decided to employ the new formulation to extend results obtained by Goldstein and Leib (JFM, 1989) for monochromatic oblique waves in compressible mixing layer. An investigation was also initiated of the parametric resonance between a 2D and an oblique wave in an incompressible mixing layer. The 2D wave description would be along the lines of a recent paper by Hultgren (in press) and the goal is to provide a more rational theory than those available at present to describe the early stages of vortex pairing.

Joseph Mathew, (Postdoctoral) MIT

The action of externally imposed, streamwise vorticity on a plane, incompressible mixing-layer is being studied with Dr. M. E. Goldstein.

Using small-amplitude expansions, it can be shown that to leading order, weak streamwise vorticity is carried unaltered by a base flow which is a mixing-layer. However, the expansion is not uniformly valid, and the next order solution grows algebraically with downstream distance. Rescaling to allow for this growth results in a nonlinear problem for the development of streamwise vorticity interacting with a vortex sheet (free-surface) boundary. The accompanying viscous problem consists of the 3D boundary-layer equations on a general surface (here, the vortex sheet). A Fourier-spectral and finite-difference scheme was used to compute the solution in the outer inviscid region, while the Keller-box method together with marching in the spanwise and streamwise directions was used for the viscous calculations. The codes were modifications of those developed by Dr. S. J. Leib.

The initial disturbance is a single sinusoid in the spanwise coordinate and corresponds to a pair of counter-rotating vortices per period. Close to the origin, the vortex-sheet displacement grows quadratically in the streamwise direction and the dominant spanwise variation is a sinusoid with twice the wavenumber of the imposed disturbance, in accordance with our analytical results. At an O(1) downstream location (scaled coordinate, $x = 0.8$) the calculations of the outer problem break down. Graphs of the spanwise derivative of the spanwise velocity at the vortex sheet and its spectrum suggest that the breakdown is due to the formation of a pair of singularities. This is not unexpected, since, in the related Kelvin-Helmholtz problem, analytic initial configurations of a vortex sheet are known to give rise to singularities at finite times. While I have tried alternate numerical strategies and local analysis, it has not been possible to establish the precise nature of the singularity in the present case. The signature of the apparent singularity in the viscous layer is the appearance of relatively higher transverse velocities at its outer edge at the same spanwise locations.

Kazuo Matsuuchi, University of Tsukuba

A well-known phenomena in transitional pipe flow is the peculiar turbulent lump called a "puff". Prior experimental work by me and my collaborators has established the necessary conditions for a puff to occur but the dynamics included in the complex flow behavior of a puff are not clear.

On the basis of our experimental results, I discussed the generating and sustaining mechanism of a puff with Dr. R. J. Deissler and others at ICOMP. After discussions and rereading Dr. Deissler's paper[1] on the generation of slugs, it was apparent that the generation of puffs could never be understood in a way similar to the case of slugs. The generation of slugs can be explained by assuming weak disturbances. Furthermore, our experiment and others[2] show that the existence of an inlet region, where linear instability may occur, is not necessary for the generation of puffs as opposed to that of slugs. As a result, it was recognized that the fluctuations of strong intensity modify the mean flow profile locally and there appears a narrow unstable region

moving with the mean velocity. This local and strong nonlinear character is essential in understanding the dynamics of a puff.

REFERENCES

Deissler, R. J.: Turbulent Bursts, Spots and Slugs in a Generalized Ginzburg-Landau Equation. Phys. Lett. A, Vol. 120, No. 7, 1987, pp. 334-340.
Epper, R.; and Fasel H., eds.: Laminar-Turbulent Transition: Symposium, Springer, 1980, pp. 17-26.

Mohammed A. Mawid (Postdoctoral), University of Illinois, Chicago

Numerical investigations of gaseous and spray laminar diffusion flames have been conducted. The numerical modeling was based on the KIVA-II computer code. The velocity, concentration, and temperature fields were predicted and compared with the classical Burke-Shumman gaseous diffusion flame and with experimental observations for the spray flame. Major differences were identified between the numerically calculated and analytic flames. The computed gaseous flame was found to be taller and thinner than the Burke-Shumman flame. Spray flames were taller than the counterpart gaseous ones due to the presence of liquid fuel droplets. The investigations further revealed a number of significant characteristics about the structures of these flames. This was the first phase of a project which aimed towards developing and testing predictive numerical capabilities and physical models related in particular to the liquid-phase.

The second phase of the project is currently underway. This phase consists of calculating a coaxial unconfined turbulent combusting spray where the experimental measurements made by Dr. D. L. Bulzan at NASA Lewis Research Center are being used as inlet conditions. Major difficulties related to implementing the KIVA-II code to predict the flow field are being encountered and remedies are being tested to overcome these difficulties. A new formulation, for example, to accelerate the attainment of a steady-state solution is being tested. The results obtained so far appear to be promising.

A. F. Messiter, University of Michigan

According to linear theory, a vortex sheet in supersonic flow is neutrally stable at sufficiently high Mach number (Miles, J. Fluid Mech., 1958). But, if a sound wave is incident upon the sheet, weak nonlinearity leads to the appearance of "kink modes" (Artola and Majda, Physica D, 1987), with discontinuities in shape such that a weak shock wave is present on one side and a weak centered expansion on the other. Similarly, one would expect that an assumed (unforced) perturbation in shape that is initially smooth would become distorted at large time. An alternative derivation, equivalent to that of Artola and Majda, has been carried out, and shows that an initially smooth small-amplitude disturbance that propagates at the neutral wave speed of linear theory will develop slope discontinuities at a specific large value of the time. The derivation is nearly completed and a report is in preparation. The ultimate goal, of course, is to relate the behavior of the vortex sheet to the behavior of a shear layer of nonzero thickness subjected to a long-wave disturbance, as in the work of T. Balsa.

I have also discussed with M. Goldstein and S. Leib certain aspects of boundary-layer distortion by free-stream vorticity, with D. Wundrow some results relating to instability of hypersonic boundary layers, and with T. Balsa his work on instability of supersonic shear layers.

Subodh K. Mital (Postdoctoral), Case Western Reserve University

Work proceeded in the area of composite microfracture driven by both mechanical and thermal loads. Both unidirectional and cross-plied composites were evaluated for microfracture and to characterize the effect of microfracture on global response. Work is also continuing in the area of hybrid composites to characterize the overall composite mechanical properties using 3D finite element models, microfracture evaluation, and comparison of microstresses with those predicted by micro- and macro-mechanics embedded in the in-house codes.

R. A. Nicolaides, Carnegie Mellon University

Adaptive solution techniques for multi-dimensional flows with boundary layers are one of my current interests. Much of the existing work is derived from elliptic partial differential equation methods which refine the mesh simultaneously in all coordinate directions. For example, a mesh cell such as a triangle in 2D would be refined into four congruent triangles in this type of refinement. This is not natural for boundary layer refinement where significant changes in the solution can occur in fewer dimensions than the dimensionality of the flow domain itself. During my visit to ICOMP I continued my work on this topic. I have developed a new approach which permits lower dimensional refining and also coarsening of the mesh. There are two main issues, which should be independent of each other. First is the question of where and when to refine the mesh. Second is the question of how to actually manage the data structures which are necessary for effective application of adaptive methods. My work at ICOMP was mainly on the second topic. An implementation is currently underway and will be reported in detail later.

Alexander Oron, Technion-Israel Institute of Technology

During my visit at ICOMP, I investigated, in collaboration with Dr. R. J. Deissler, stable steady localized structures emerging on the free surface of a thin liquid film bounded by a horizontal plate from above and open to the atmosphere from below and experiencing surface traction arising from the surface tension dependence on temperature (the Marangoni effect). In the given situation, gravity destabilizes the free surface (Rayleigh-Taylor instability) while surface tension stabilizes it. Our studies showed that, in the case of the plate cooler than the ambient air, the Marangoni effect stabilizes the fluid-air interface and in conjunction with surface tension can prevent the liquid film from rupture leading to the emergence of stable steady localized patterns. The investigation was based on a study of a 2D asymptotic equation describing the spatio-temporal evolution of the free surface of the liquid film derived in references one and two. We showed that the equation has a Liapunov functional that is nonincreasing in time and using this property we demonstrated the existence of stable axisymmetric localized patterns.

The Marangoni effect is found to be very important in a low gravity environment and, therefore, for processes of space technology, it will be very interesting to see whether a thin liquid film on the underside of a cooled horizontal plate will exhibit such stable stalactite-like structures.

REFERENCES

.Kopbosynov, B. K.; and Pukhnachev, V. V.: Thermocapillary Flow in Thin Liquid Films. Fluid Mech.-Sov. Res., vol. 15, no. 1, 1986, pp. 95-106.

Oron, A.; and Rosenau, P.: Formation of Patterns Induced by Thermocapillarity and Gravity. T. Phys. II France, vol. 2, 1992, pp. 131-146.

Christophe Pierre, University of Michigan

The basic purpose of this research is the development of computational methods for understanding and predicting the effects of unavoidable blade-to-blade dissimilarities, or mistuning, on the dynamics of nearly cyclic bladed-disk assemblies. This topic is of importance as mistuning has been shown to increase the forced response amplitudes of some blades significantly, and even to lead to blade failure. Furthermore, the current trend toward high performance propulsion turbomachinery designed for finite service life demands an accurate prediction of system performance and dynamics.

We have examined the effects of mistuning on the aeroelastic modal characteristics of blade assemblies [1]. We have shown that the aeroelastic response of blade assemblies is *highly sensitive* to a small mistuning. In particular, the aeroelastic modes of the system lose their constant interblade phase angle characteristic when mistuning is introduced: they become severely *localized* to a few blades and no pattern can be discerned for the interblade phase angle. Furthermore, the root locus of the aeroelastic eigenvalues loses the regular pattern that characterizes the tuned system to become apparently randomly scattered for small mistuning. We have developed perturbation schemes that explain and predict this sensitivity to mistuning.

We have applied the general findings of reference [1] to the model of an actual mistuned high-energy turbine, namely the first stage of turbine blades of the oxidizer turbopump in the space shuttle main rocket

engine [2] Our results show an *extreme* sensitivity to mistuning for the aeroelastic modes of the turbine. The transition from constant-interblade-phase-angle modes to localized modes is very rapid and exhibits a complex behavior of the eigensolution. Severe localization occurs for blade frequency mistuning of approximately 0.1%--a mistuning level clearly unavoidable for a real turbine. Contrary to previously suggested failure mechanisms (*e.g.*, thermal shock), our work proposes a theory which is based on the intrinsic dynamic characteristics of mistuned rotors.

We have developed a transfer matrix methodology that allows us to cast the dynamics of blade assemblies in terms of the coupling coordinates between blades, which has conceptual as well as computational advantages. A paper describing these results is nearly ready [3].

Finally, we have initiated research on the dynamics of mistuned assemblies with *several* component modes per blade. Namely, the interaction of two close blade modes and the resulting effect on the sensitivity to mistuning are being examined. We are also beginning to utilize our free response findings for the study of the forced response problem.

PUBLICATIONS

Murthy, D. V.; and Pierre, C.: Aeroelastic Modal Characteristics of Mistuned Blade Assemalies: Mode Localization and Loss of Eigenstructure. NASA TM-104519, 1991 (Also, AIAA Paper 91-1218, 1991).

Murthy, D. V.; Pierre, C.; and Smith, T. E.: Localization of Aeroelastic Modes in Mistuned High-Energy Turbines. NASA TM-104445;, 1991 (Also, AIAA Paper 91-3379, 1991).

Ottarsson, G.; and Pierre, C.: A Transfer Matrix Approach to Vibration Localization in Mistuned Blade Assemblies. Prepared for the 1992 ASME International Gas Turbine and Aeroengine Congress and Exposition, Cologne, Germany.

R. H. Pletcher, Iowa State University

Work continued with Philip Jorgenson on the simulation of internal viscous flows using unstructured grids. The objective of the present study is to develop an accurate and economical simulation strategy for the solution of the Navier-Stokes equations utilizing unstructured grids that is particularly well suited for the turbomachinery applications. The scheme will be applicable to flows over a wide range of Mach numbers from the incompressible limit through transonic. Progress has been made in most key facets of the study. A grid generation scheme based on triangulation has been developed in two dimensions. Validation of the implicit numerical formulation is in progress. Work is also underway to compare several candidate sparse matrix solvers for use with unstructured grids.

Some time was also devoted to obtaining a better understanding of the cause of the low Mach number limitation observed in most numerical schemes for computing compressible flows. It was found that by adding a properly formulated pseudo time term to each conservation equation, the convergence rate could be made virtually independent of Mach number for steady flows. It also appears that this procedure can be made time accurate, although work is still underway for unsteady flows. This study on low Mach number flows uses the same implicit primitive variable strategy being employed in the unstructured grid work but the demonstration calculations were carried out with a scheme formulated on structured grids.

Aamir Shabbir, (Postdoctoral) SUNY, Buffalo

1. Turbulence: Part of my effort was directed toward evaluation and development of turbulence models. The usual approach in establishing the correctness and accuracy of these models is to numerically solve the modeled differential equations and then compare the results with the experiment. However, in the case of a discrepancy, this procedure does not pinpoint where in the model lies the drawback. It is also possible that the model overcompensates one physical phenomenon and undercompensates another so that the net result is a good agreement between the two. Therefore, a more desirable approach is to directly compare the individual terms in the equations with their models.

With this in mind, a direct comparison between the pressure strain and pressure temperature gradient correlations and their closure models has been carried out. The flows used included both physical and numerical experiments on homogeneous shear flows and physical experiments on buoyant plumes. Models considered included both the linear and the more elaborate non-linear ones. It is found that the non-linear

models provide a much better agreement with these experiments than the linear ones. A new model for the slow part of the pressure temperature-gradient correlation was also derived using joint realizability concept. This work was done in collaboration with Dr. T-H. Shih.

Some of my work on turbulence was concerned with experimental budgets for second moments. Despite the large volume of work on second moment closure, there is very little experimental information available about the budgets of second moments. Part of this reason stems from our inability, at present, to measure the pressure correlations. Experimental budgets for Reynolds stresses and heat fluxes had been carried out for a boundary free shear flow (round plume) and the pressure correlations were obtained as the closing terms in these budgets. These budgets show how different terms in the equations are distributed across the flow and can be used to analyze some of the modeling assumptions. For example they show that the assumption of local equilibrium is not justified for bulk of the flow field - an idea fundamental to the algebraic stress models.

2. Bypass Transition: A direct simulation of bypass transition (due to elevated free-stream turbulence) over a flat plate is being carried out using the spectral element method. The purpose of this study is to develop a better physical understanding of the phenomena and also to provide a data base for the modeling which is concurrently being pursued at CMOTT. The work is at a preliminary stage and is being done in collaboration with Dr. T-H. Shih and Professor G. Karniadakis at Princeton University.

3. Modeling of Turbomachinery Flows: Modeling of turbomachinery flows using the average passage approach is being pursued. Turbomachinery flows are turbulent and unsteady and numerical calculation of a flow in a multistage machine, at present, is not feasible. However, the effects of periodic unsteadiness can be accounted for through the models for deterministic stresses which arise in the average passage equation set (Adamczyk 1985). Exact equations governing the transport of these stresses have been derived and a two equation model is being developed and tested at present. The model uses ideas from turbulence modeling such as the gradient diffusion type hypotheses. This work is being performed in collaboration with Dr. J. Adamczyk at the NASA Lewis Research Center.

T.-H. Shih, Stanford University (CTR)

In order to identify the state of the art in engineering turbulence modeling, a critical comparison of two-equation models and second-order closure models has been performed. Twelve two-equation models, including K-ε, K-ω , q-ω and k-τ models, are implemented in a common flow solver. Calculations are carried out for 2D fully developed channel and boundary layer flows and compared with the results of direct numerical simulations and experimental data. This work was done by N. J. Lang and myself (see NASA TM 105237, ICOMP-91-15; CMOTT-91-05). For the second-order closure model the velocity-pressure gradient (and scalar-pressure gradient) tensor and the dissipation rate tensor are the two of the most important terms in the second moment transport equations. These correlation tensors are usually decomposed into a so called rapid term and a return-to-isotropy term. Five representative rapid models and eight return-to-isotropy models are examined. The DNS data of forty five homogeneous flows are used for the direct comparisons to reveal the individual behavior of the proposed models (see NASA TM 105351, ICOMP-91-25; CMOTT-91-10).

In modeling compressible turbulent flows, a two-scale model is proposed to account for the effects of the compressibility and eddy-shocklets. This is an on-going research project carried out by Dr. W. W. Liou and myself.

In modeling bypass-transition related to free-stream disturbances, an eddy-viscosity two-equation model with intermittency effect is proposed. The preliminary results show very good agreement with measurements on flat plate boundary layers. Further tests in the cases with pressure gradients are on the way. This research is performed by Dr. Z. Yang and myself. In addition, a second-order closure model is being developed for both near-wall turbulence and bypass transition.

To study the physics of bypass transition and the phenomena of the eddy-shocklets in compressible flows and to build up the necessary data base for model development, a direct numerical simulation program has been formed. Dr. A. Shabbir, Dr. A. Hsu and myself are involved in these studies.

Avram Sidi, Technion-Israel Institute of Technology

The application of vector extrapolation methods to very large and sparse linear systems of equations was investigated. Several theoretical results were proved that show that when extrapolation is performed following a number of iterations with a matrix iterative technique, convergence rates can be enhanced dramatically.

Theoretical upper bounds on the rates of convergence were derived and shown numerically to be very tight. The theoretical results have been supported with quite a few numerical examples. Based on these results, the cycling strategy is now modified to include a fixed number of iterations prior to extrapolation in each cycle. This new strategy has been observed to be vastly superior to the Arnoldi method and GMRES.

The computer program that was published in the ICOMP Report #90-20 (recently appeared in the Journal of Computational and Applied Mathematics) that implements the minimal polynomial and reduced rank extrapolation methods was modified to enable the user to invoke the SSD without increasing the core memory requirements. These methods are used in the solution of very large linear and nonlinear systems in conjunction with a fixed point iterative technique.

Joseph L. Steger, University of California, Davis

One of the factors inhibiting the more widespread use of chimera schemes in computational fluid dynamics is the lack of available software that automatically connects overset meshes together and locates grid hole points caused by other boundaries. Only the Pegasus code developed at CALSPAN and the CompGrd code of Brown, Henshaw, and Chesshire are currently available, although Meakin of U.C. Davis is developing a third code. Under the assumption that the availability of more and better connectivity codes will make chimera schemes less imposing to potentially new users, during my stay at ICOMP I undertook the development of yet another connectivity code, albeit a simple 2D code, which also tests another kind of searching mechanism.

The new connective code is intended to test logic that will ultimately go into a 3D code in which grids are radiated off of a body surface that has been blanketed with overset surface meshes. To hasten searches in the field, all curvilinear grid points are "bucketed" within the cells of a reference Cartesean background grid. To find curvilinear grid points for interpolating boundary values for another curvilinear grid, one simply searches the "bucket" that contains the boundary point that is to be interpolated. Even if no other curvilinear grid points are contained within the bucket, the index of a nearby point is stored in the bucket. The new connectivity code was programmed and checked out to this level, and the searches appear to run much more efficiently than those used in the standard Pegasus code. Additional special logic for searching fine viscous grids near body surfaces has also been coded but not yet checked out.

Mark Stewart, (Postdoctoral) Princeton University

Research during the last year has concentrated on developing grid generation and numerical solution techniques and applying them to jet engine configurations. To demonstrate the use of multi-block grid generation techniques, a series of solutions for an engine configuration based on the General Electric Energy Efficient Engine (E^3) was developed. First, a 2D non-axisymmetric Euler solution was developed for a full engine configuration including external flow and the internal flow through the inlet, fan duct, core duct and nozzle. The effects of the blades and combustor are not included in this simulation as a simplification. This model was extended to an axisymmetric Euler solution which represents the flow through an engine with no blades or combustor. This series of simulations is being developed further by implementing models of blades and a combustor.

The multi-block Euler solver used in these calculations was ported to an Intel Hypercube iPSC/3 in a joint effort with Rodger Dyson. This work is concentrating on the techniques necessary to implement a CFD code on a parallel machine, the use of multiblock grids for load balancing and the efficiency of the resulting code.

T. W. Swafford, Mississippi State University

Two areas associated with ongoing aeroelasticity research were investigated. The first was with regard to the appearance of total pressure losses in current versions of Whitfield's flux-difference-split two-pass scheme for solving the Euler equations. It was speculated that a possible cause of the apparent loss in stagnation pressure was grid-related (the same phenomena has been observed for similar but unrelated grids at Mississippi State). Consequently, a different version of the code was fabricated which uses C-grids as opposed to H-type grids which are currently employed in the aeroelasticity efforts. Unfortunately, similar trends regarding total pressure changes near solid surfaces were observed with the C-grid version as well. It was concluded that this anomaly has its origin elsewhere, perhaps in boundary condition specification.

The second effort centered around building a multiblock version of the coupled, aerostructures code (for

use by T.S.R. Reddy) such that nonzero interblade phase angle cases can be accommodated (current versions employ only one blade passage which will suffice for zero interblade phase angle). A two-blade-passage (or two block) version has been put together and steady-state comparisons between results generated by the one- and two-block codes for the F21 propfan are dangerously favorable (there is nothing more dangerous than answers that look about right). This effort will continue concurrently with that associated with the single-block, coupled version.

Gretar Tryggvason, University of Michigan

Work continued on the development and application of the Tracked, Immerged Boundary (TIB) technique to simulate multifluid flows. The basic method was developed by S. O. Unverdi in his doctoral work and is described in Unverdi and Tryggvason: A Front Tracking Method for Incompressible Flows (to appear in J. Comput. Phys.). Some time was devoted to rewriting the code that resulted from Unverdi's thesis. In addition to general improvement in program structure, several major changes were made, including increasing the accuracy of the time integration to second order and modification of the calculation of the mean curvature. These changes resulted in a program that is less than half the length of the original program and more efficient. Investigations of droplet collision, using this technique, were initiated in collaboration with D.Jaqmin. A technique to accelerate the drops towards each other by applying a nonuniform body force was developed and tested and a code to collect various diagnostics was implemented. I also worked on visualization of the results and Don Sosoka made a video animation of colliding drops as computed by the TIB technique.

Eli Turkel, Tel Aviv University, Israel

Work continued with P. Jorgenson of NASA Lewis on the use of central difference schemes for time dependent problems with shocks. The schemes are all advanced using a Runge-Kutta formula in time. Near shocks a second difference is added as an artificial viscosity. This reduces the scheme to a first order upwind scheme at shocks. The switch that is used guarantees that the scheme is locally TVD. For steady state problems it is usually advantageous to relax this condition. Then small oscillations do not activate the switches and the convergence to a steady state is improved.

To sharpen the shocks different coefficients are needed for different equations and so a matrix valued dissipation is introduced and compared with the scalar viscosity. The connection between this artificial viscosity and flux limiters is shown. Any flux limiter can be used as the basis of a shock detector for an artificial viscosity. We compare the use of the Van Leer, Van Albada, Minmod, Superbee and the "average" flux limiters for this central difference scheme. For time dependent problems we need to use a small enough time step so that the CFL was less than one even though the scheme was linearly stable for larger time steps. Using a TVB Runge-Kutta scheme yields minor improvements in the accuracy.

Work has begun on adding chemistry terms to the multistage Navier-Stokes solver that has been developed over the past few years.

A new implicit solver for spectral methods is being developed. It is based on an LU decomposition of the Chebyshev derivative matrix. Preliminary runs for the 1D Euler equations show high convergence rates to the steady state. Extensions to two space dimensions are under development.

William J. Usab, Jr., Purdue University

The objective of the present research is the development of an improved multigrid acceleration scheme for the solution of steady viscous flow problems. It has been observed that the application of current explicit time-marching schemes to problems involving highly stretched meshes results in very poor overall convergence rates, even with multigrid acceleration. Since highly stretched meshes are often used in the solution of viscous flow problems this is a critical issue in the application of these methods to complex 3D flow problems. The source of the problem lies in extremely small time-steps which must be used for stability in fine mesh regions. Even with local time-stepping overall convergence is very poor due to the very slow change in the steady-state residuals in these regions. Further, local time-stepping on highly stretched meshes distorts the time-evolution of the solution, resulting in an unsteady problem governed by a set of partial differential equations which are different from the original unsteady Navier-Stokes equations. While the steady-state solution remains the

same with or without local time-stepping, understanding how the governing equations change may lead to a better time-marching scheme for these types of problems. The present research is being pursued with Dr. John Adamczyk and Mr. Mark Celestina here at the NASA Lewis Research Center.

To gain a better understanding of the "time" evolution of the solution when local time-stepping is performed on highly stretched meshes the following simple model problem was formulated. Consider the solution of the Euler equations on an orthogonal mesh with uniform spacing in x and a variable, but prescribed spacing in y. The governing equations may be transformed to computational space, (ξ,η) where the corresponding transformed mesh is uniform. Next a new transformed time variable, T, can be defined based on the local time-stepping criteria for which a constant time-step in the transformed variable is equivalent to a local time-step based on the CFL condition for a given scheme. The resulting transformed equations may be viewed as a set of partial differential equations which describe the "time" evolution of a different unsteady problem in (ξ,η,T). In particular, these equations and the resulting "time" evolution are a function of the prescribed mapping between physical and computational space, and the time stepping criteria used.

If one now considers a solution of these equations which is a small perturbation about a uniform flow in the x-direction, these transformed equations can be reduced to a single linear third-order PDE for the perturbation pressure. One can also derive equations for each of the other perturbation variables, if desired. (This analysis is very similar to that used in the study of unsteady motion on transversely sheared mean flows by Goldstein [Ref. 1]). With specification of an initial perturbation and the coordinate mapping, these equations can, in principle, be solved for the time evolution of the solution with local time scaling. Thus, showing how mesh stretching and in turn local time-stepping effects the evolution of the exact solution. Even without directly solving these equations through examination of the coefficients in the PDE, insight can be gained in how mesh stretching effects the propagation of pressure disturbances as the solution develops. With this information, it may be possible to construct a better implicit residual averaging operator or possibly a new time marching algorithm which speeds the propagation of errors out of the fine mesh regions of the domain.

To date, the PDE for the perturbation pressure has been derived. Two types of mesh stretching in the y-direction were then considered; an algebraic stretching often used in laminar flow problems, and a hyperbolic tangent stretching commonly used in turbulent flow problems. In both cases, it is clear that the dominant effect on the time evolution of the solution is due to the local aspect ratio of the mesh, with the rate of change of aspect ratio playing a minor role. Further, as might be expected, in regions where the mesh aspect ratio is very large, there is only a very weak dependence on the variation of the perturbation pressure in the streamwise direction. Finally, the resulting equation for the perturbation pressure does not have any singular points as a result of these prescribed mesh stretchings. Based on the character of these equations, it may only be possible to accelerate the convergence to steady-state by moving to a more implicit formulation. Short of dropping the use of an explicit time marching algorithm in fine mesh regions, acceleration may be possible with a new residual averaging operator or by adopting a locally implicit formulation in the direction of high mesh stretching.

In addition to the above research, an ICOMP seminar entitled, "A Solution Adaptive Unstructured Scheme for Quasi-3D Flows Through Advanced Turbomachinery Cascades," was given. Also, during this period, working with Yitsann Jiang (one of my graduate students from Purdue) we incorporated the necessary boundary conditions into this adaptive unstructured code to allow solution of cascade flow problems with supersonic axial inflow and outflow. Adaptive unstructured solutions for a 2D cascade of double wedge airfoils were computed for two different supersonic operating conditions (AGARD test CASE A/CA-2 [Ref. 2]). The computed solutions were shown to be in excellent agreement with the theoretical solutions obtained using shock-expansion theory. This work is currently being supported at Purdue University by Dr. David Miller through a NASA Lewis grant.

REFERENCES

Goldstein, M. E.: Characteristics of the Unsteady Motion on Transversely Sheared Mean Flows . J. Fluid Mech., vol. 84, Jan. 1987, pp. 305-329.

Fottner, L.: Test Cases for Computation of Internal Flows in Aero Engine Components. AGARD AR-275, 1990.

Bram Van Leer, University of Michigan

William Colrier (CFD Branch) and I made an extensive stability and monotonicity analysis of flux-vector splitting, which subsequently was made into a NASA TM (#1004353), and was presented at the AIAA 10th

CFD Conference in Honolulu, Hawaii, June 1991 (AIAA-91-1566). We demonstrated it is not possible to make a robust and non-oscillatory flux-vector splitting that has vanishing diffusion across a boundary layer.My undergraduate student, Eric Charlton, (University of Michigan) and I collaborated with Hung Huynh (CFD Branch) on the development of non-oscillatory extremum-preserving higher-order advection methods.

J. J. W. Van Der Vegt, Stanford University (CTR)

Transition to turbulence in practical aerospace applications usually occurs in a strongly disturbed environment. For instance, the effects of freestream turbulence, roughness and obstacles in the boundary layer strongly influence transition. Proper understanding of the mechanisms leading to transition is crucial in the design of aircraft wings and gas turbine blades, because lift, drag and heat transfer strongly depend on the state of the boundary layer, laminar or turbulent. Unfortunately, most of the transition research, both theoretical and experimental, has focused on natural transition. Many practical flows, however, defy any theoretical analysis and are extremely difficult to measure. This project aims at studying the effects of free stream turbulence at high subsonic and transonic Mach numbers by means of direct simulations. This is especially important in a gas turbine environment, where turbulence intensities are measured between 5 and 20%. Direct numerical simulations offer a unique opportunity to study specific phenomena while excluding disturbances from other sources. This research will provide data which can be used to clarify mechanisms leading to transition in an environment with high free stream turbulence levels. This information is useful for the development of turbulence models which are of great importance for CFD applications and are currently unreliable for more complex flows such as transition.

A fully implicit and time accurate higher order implicit finite volume scheme is currently being developed. The Steger-Warming and Osher flux splitting are used in the upwind spatial discretization and a Newton-Raphson iteration is used to obtain time accuracy. The numerical scheme is designed to have both a high accuracy in boundary layers and the ability to capture shocks. The first numerical simulation will investigate the effect of free stream turbulence on the transition at a flat plate at a Mach number of about .85. The level of free stream turbulence at the leading edge will be approximately 5%. These conditions are representative for flow conditions of a turbine blade.

Dave Whitfield, Mississippi State University

In an effort to accelerate convergence and perhaps increase the time step limit of some iterative schemes for solving the Euler and Navier-Stokes equations, various forms of the conjugate gradient method [1] were investigated. The methods considered were the preconditioned conjugate gradient (CG), and the preconditioned and nonpreconditioned conjugate gradient-squared (CGS) method [2,3]. The preconditioner was one of the solution matrices used in [4] to solve the Euler or Navier-Stokes equations. This preconditioner can be viewed [4] as a symmetric Gauss-Seidel scheme without residual updating between forward and backward passes. It can also be viewed as a factored scheme.

The test case used for these computations was the numerical solution of the 2D Euler equations for flow over a cylinder. The scheme is upwind (Roe), finite volume, second order in space, and first order in time. Results from this investigation must be considered preliminary, but the following was observed.

1. The preconditioned CGS method converged at each time step much faster than the preconditioned CG method. For example, after five or six cycles the CGS method converged eight to nine orders of magnitude, whereas, the CG method converged only about three or four orders of magnitude and usually did not get below five orders of magnitude even after 15 or 20 cycles. (Note that this is all at one time step of the Euler solution.) However, a converged steady-state Euler solution could be globally converged to about the same order of magnitude using four or five cycles of either the CG or the CGS method in the same number of total cycles. (Note that total cycles here corresponds to the complete converged solution of the Euler equations.) In this sense, the CG method is better because it has a smaller operation count (by a factor of two as coded now) than the CGS method.
2. Good preconditioning was essential.
3. No significant increase in time step could be obtained for either the CG or CGS method compared to the symmetric Gauss-Seidel method.
4. After all this, the winner of convergence rate versus CPU time on a single processor scalar machine, was the symmetric Gauss-Seidel method.

REFERENCES

Hestenes, M. R.; and Stiefel, E.: Methods of Conjugate Gradients for Solving Linear Systems. Nat. Bur. Stand., J. Res., vol. 49; 1954, pp. 409-436.
Sonneveld, P.: CGS, A Fast Lanczos-Type Solver for Nonsymmetric Linear Systems. SIAM J. Sci. Statist. Comput., vol. 10, no. 1, 1989, pp. 36-52.
Dongarra, J. J. et al.: Solving Linear Systems on Vector and Shared Memory Computers. Society for Industrial and Applied Mathematics, Philadelphia, 1991.
Whitfield, D. L.; and Taylor, L. K.: Discretized Newton-Relaxation Solution of High Resolution Flux-Difference Split Schemes. AIAA Paper 91-1539, 1991.

Zhigang Yang, (Postdoctoral) Cornell University

During the past year, I worked on modeling of near wall turbulence and modeling of bypass transition due to freestream turbulence. Both projects were done in collaboration with Dr. T. H. Shih of ICOMP.

Traditionally, turbulence models are devised for high turbulent Reynolds number flows and are used in conjunction with a wall function when applied to wall bounded turbulent flows. Since universal wall functions do not exist in complicated flow situations, it is necessary to develop turbulence models for near wall turbulence. We proposed a κ-ε model for near wall turbulence. In this model, the turbulent velocity scale and turbulent time scale are used to define the eddy viscosity. The time scale is shown to be bounded from below by the Kolmogorov time scale. The dissipation equation is reformulated using this time scale, removing the need to introduce pseudo-dissipation. A damping function is chosen such that the shear stress satisfies near wall asymptotic behavior. The model constants used are the same as those in the commonly used high turbulent Reynolds number κ-ε model. Thus, when it is far away from the wall, the proposed model reduces to the standard κ-ε model. Fully developed turbulent channel flows and turbulent boundary layer flows over a flat plate at various Reynolds numbers are used to validate the model. The model predictions are found to be in good agreement with the DNS data. This work was presented at the 4th International Symposium on Computational Fluid Dynamics, September, 1991.

In a quiescent environment, transition is initiated by the amplification of Tollmien-Schlichting waves. These waves eventually breakdown, giving rise to turbulent spots, which are precursors of the turbulent boundary layer. While in an environment with high freestream turbulence, say the flow passing over a turbine blade, turbulent spots are formed due to the transport of turbulence from the freestream to the boundary layer rather than from T-S wave amplification. Thus the transition follows a bypass scenario. We used a low Reynolds number κ-ε model to calculate the bypass transition due to the freestream turbulence. The transitional boundary layer over a flat plate with different freestream turbulence levels was taken as a test case. It was found that the model could mimic the transitional flow. However, the predicted transition was found to be sensitive to the initial conditions. The paper describing this work will appear in Proceedings of ICASE/LaRC Workshop on Transition and Turbulence, to be published in Spring 1992 by Springer-Verlag.

A. Yoshizawa, University of Tokyo

Two subjects were addressed during my visit at ICOMP. One was the investigation of the general properties of helicity. The importance of helicity has been discussed in several phenomena related to engineering and natural sciences. Another was the derivation of a Reynolds stress model equation using the results of a two-scale direct interaction approximation (DIA). The renormalization of the representation of eddy-viscosity type for the Reynolds-stress leads to an equation similar to other Reynolds stress model equations. The feature of this derivation lies in the fact that no direct use of the pressure-strain term is made and that some additional effects like helicity can be easily incorporated using the corresponding eddy-viscosity-type representation for the Reynolds stress.

Shayne Yungster, (Postdoctoral) University of Washington

Three major CFD research efforts were carried out at ICOMP during the last year. These are summarized below.

Ram Accelerator: This task consisted of a computational study of the superdetonative propulsion modes of a ramjet-in-tube projectile accelerator concept known as the ram accelerator. Several configurations at various flight conditions were investigated in a Mach number range of $5 < M < 9$. The investigation was carried out using a new fully implicit total variation diminishing (TVD) code that solves the Reynolds-averaged Navier-Stokes equations including finite-rate chemistry. The combustion model used consisted of a 7-species/8-step reaction mechanism. Using this CFD code, the performance characteristics (i.e. thrust, viscous drag, ballistic efficiency, etc.) of various ram accelerator configurations were studied. This work included also a study of shock-wave/boundary layer interactions involving combustible gases, spontaneous ignition in the boundary layer, and the stability of the shock-induced combustion process. This work is detailed in ICOMP Report No. 90-22 (submitted for publication in the AIAA Journal) and ICOMP Report No. 91-10.

CFD Evaluation of External Burning: A CFD effort in support of experimental studies (being conducted at LeRC) aimed at developing methods for improving low-and-high-speed nozzle performance, and to demonstrate transonic drag reduction benefits of external burning for NASP type vehicles has been started. This task is utilizing the above mentioned TVD code and also three-dimensional, multiblock versions of the RPLUS code. Results obtained so far include two-dimensional calculations of laminar and turbulent flow, with and without external burning. Initially, the external fuel injection process has been simplified and modeled as a fuel/air stream whose properties (such as jet penetration, equivalence ratio, etc.), have been determined from experimental data. These simplified calculations will help determine the effects of external combustion on nozzle performance.

Presently, 3D calculations of the nozzle and external flow are being performed with the RPLUS code. Subsequent calculations will use detailed models of the injection region. The CFD studies will include simulations of large scale flame holding/ignition experiments, and hot/cold nozzle tests of current NASP configurations.

Oblique Detonation Wave Studies: The main objective of this study was to compare numerical and asymptotic solutions of oblique detonation waves, assuming a simple finite-rate combustion model. Numerical solutions obtained with a TVD scheme and with the RPLUS code were compared with closed form asymptotic solutions obtained with the method developed by Joseph Powers (University of Notre Dame). It was determined that for sufficiently high Mach numbers and heat release, the results obtained with the asymptotic and numerical approaches are in very good agreement. The results of this study will be presented at the Fourth International Conference on Numerical Combustion, St. Petersburg, FL, December 2-4, 1991.

REPORTS AND ABSTRACTS

The ICOMP Steering Committee: "Institute for Computational Mechanics in Propulsion (ICOMP), Fifth Annual Report-1990," ICOMP Report No. 91-1, NASA TM-103790, May 1991, 48 pages.

The Institute for Computational Mechanics in Propulsion (ICOMP) is operated jointly by Case Western Reserve University and the NASA Lewis Research Center in Cleveland, Ohio. The purpose of ICOMP is to develop techniques to improve problem-solving capabilities in all aspects of computational mechanics related to propulsion. This report describes the activities at ICOMP during 1990.

Gupta, Murli M. (ICOMP): "Spectrum Transformation for Divergent Iterations," ICOMP Report No. 91-2, NASA TM-103745, March 1991, 18 pages.

In this paper we describe certain spectrum transformation techniques that can be used to transform a diverging iteration into a converging one. We consider two techniques called spectrum scaling and specturm enveloping and discuss how to obtain the optimum values of the trans-formation parameters. Numerical examples are given to show how this technique can be used to transform diverging iterations into converg-ing ones; this technique can also be used to accelerate the converg-ence of otherwise convergent iterations.

Jiang, Bo-nan (ICOMP): "The L_1 Finite Element Method for Pure Convection Problems," ICOMP Report No. 91-3, NASA TM-103773, April 1991, 28 pages.

In this paper we first introduce the least-squares (L_2) finite element method for two-dimensional steady-state pure convection problems with smooth solutions. We prove that the L_2 method has the same stability estimate as the original equation, that is, the L_2 method has better control of the streamline derivative. Numerical convergence rates are given to show that the L_2 method is almost optimal. Then we use this L_2 method as a framework to develop an iteratively reweighted L_2 finite element method to obtain a least absolute residual (L_1) solution for problems with discontinuous solutions. This L_1 finite element method produces a non-oscillatory, non-diffusive and highly accurate numerical solution that has a sharp discontinuity in one element on both coarse and fine meshes. We also devise a robust reweighting strategy to obtain the L_1 solution in a few iterations. A number of examples solved by using triangle and bilinear elements are presented.

Ahn, Kyung H. (ICOMP) and Ibrahim, Mounir B. (Cleveland State University): "A 2D Oscillating Flow Analysis in Stirling Engine Heat Exchangers," ICOMP Report No. 91-4, NASA TM-103781, June 1991, 7 pages.

A 2D oscillating flow analysis has been conducted simulating the gas flow inside Stirling engine heat exchangers. Both laminar and turbulent oscillating pipe flow has been investigated numerically for Re_{max}=1920 (Va=80), 10800 (Va=272), 19300 (Va=272), 60800 (Va=126). The results are compared with experimental results of previous investigators. Also predictions of the flow regime on present oscillating flow conditions have been checked by comparing velocity amplitudes and phase difference with those from laminar theory and quasi-steady profile. A high Reynolds number k-ε turbulence model was used for turbulent oscillating pipe flow. Finally, performance evaluation of the k-ε model was made to explore the applicability of quasi-steady turbulent models to unsteady oscillating flow analysis.

Demuren, Ayodeji O. (ICOMP): "Characteristics of 3D Turbulent Jets in Crossflow," ICOMP Report No. 91-5, NASA TM-104337, April 1991, 22 pages.

Three-dimensional turbulent jets in crossflow at a low to medium jet-to-crossflow velocity ratios are computed with a finite-volume numerical procedure which utilizes a second-moment closure model to approximate the Reynolds stresses. A multigrid method is used to accelerate the convergence rate of the procedure. Comparison of the computations to measured data show good qualitative agreement.

All trends are correctly predicted, though there is some uncertainty on the height of penetration of the jet. The evolution of the vorticity field is used to explore the jet-crossflow interaction.

Michelassi, Vittorio (ICOMP) and Shih, Tsan-Hsing (ICOMP): "Low Reynolds Number Two-Equation Modeling of Turbulent Flows," ICOMP Report No. 91-6, CMOTT 91-1, NASA TM-104368, May 1991, 24 pages.

A new κ-ε turbulence model that accounts for viscous and wall effects is presented. The proposed formulation does not contain the local wall distance thereby making very simple the application to complex geometries. The formulation is based on an existing κ-ε model that proved to fit very well with the results of direct numerical simulation. The new form is compared with nine different two-equation models and with direct numerical simulation for a fully developed channel flow at $Re = 3300$. The simple flow configuration allows a comparison free from numerical inaccuracies. The computed results prove that few of the considered forms exhibit a satisfactory agreement with the channel flow data. The new model shows an improvement with respect to the existing formulations.

Shih, Tsan-Hsing (ICOMP), Chen, J.-Y. (Sandia National Labs) and Lumley, John L. (Cornell University): "Second Order Modeling of Boundary-Free Turbulent Shear Flows," ICOMP Report No. 91-7, CMOTT 91-2, NASA TM-104369, May 1991, 16 pages.

This paper presents a set of realizable second order models for boundary free turbulent flows. The constraints on second order models based on the realizability principle are reexamined. The rapid terms in the pressure correlations for both the Reynolds stress and the passive scalar flux equations are constructed to exactly satisfy the joint realizability. All other model terms (return-to-isotropy, third moments and terms in the dissipation equations) already satisfy realizability (Lumley 1978, Shih and Lumley 1986). To correct the spreading rate of the axisymmetric jet, an extra term is added to the dissipation equation which accounts for the effect of mean vortex stretching on dissipation. The test flows used in this study are the mixing shear layer, plane jet, axisymmetric jet and plane wake. The numerical solutions show that the new unified model equations (with unchanged model constants) predict all these flows reasonably as the results compare well with the measurements. We expect that these model equations would be suitable for more complex and critical flows.

Mital, Subodh K. (ICOMP) and Chamis, Christos C. (NASA Lewis): "Microfracture in High Temperature Metal Matrix Crossply Laminates," ICOMP Report No. 91-8, NASA TM-104381, December 1991, 12 pages.

Microfracture (fiber/matrix fracture, interphase debonding and inter-ply delamination) in high temperature metal matrix composites (HTMMC), subjected to both mechanical and thermal loading, is computationally simulated. A crossply 0.3 fiber volume ratio SiC/Ti15 composite with 0/90/0 lay-up is evaluated for microfracture using a multicell finite element model. A computational simulation procedure based on strain energy release rates is used to predict the fracture process and establish the hierarchy of fracture modes. Microfracture results for various loading cases are presented and discussed.

Shabbir, Aamir (ICOMP), Shih, Tsan-Hsing (ICOMP) and Lumley, John L. (Cornell University): "Advances in Modeling the Pressure Correlation Terms in the Second Moment Equation," ICOMP Report No. 91-9, CMOTT 91-3, NASA TM-104413, June 1991, 36 pages.

In developing turbulence models, different authors have proposed various model constraints in an attempt to make the model equations more general (or universal). The most recent of these are the realizability principle (Lumley 1978, Schumann 1977), the linearity principle (Pope 1983), the rapid distortion theory (Reynolds 1987) and the material indifference principle (Speziale 1983). In this paper we will discuss several issues concerning these principles and will pay special attention to the realizability principle raised by Lumley (1978). Realizability (defined as the requirement of non-negative energy and Schwarz' inequality between any fluctuating quantities) is the basic physical and mathematical principle that any modeled equation should obey. Hence, it is the most universal, important and also the minimal requirement for a model equation to prevent it from producing unphysical

results. In this paper we will describe in detail the principle of realizability, derive the realizability conditions for various turbulence models, and propose the model forms for the pressure correlation terms in the second moment equations. Detailed comparisons of various turbulence models (Launder et al. 1975, Craft et al. 1989, Zeman and Lumley 1976, Shih and Lumley 1985 and one proposed here) with experiments and direct numerical simulations will be presented. As a special case of turbulence, we will also discuss the 2D two-component turbulence modeling.

Yungster, Shaye (ICOMP): "Navier-Stokes Simulation of the Supersonic Combustion Flowfield in a Ram Accelerator," ICOMP Report No. 91-10, NASA TM 104439, AIAA 91-1916, 26 pages.

A computational study of the ram accelerator, a ramjet-in-tube device for accelerating projectiles to ultra-high velocities, is presented. The analysis is carried out using a fully implicit TVD scheme that efficiently solves the Reynolds-averaged Navier-Stokes equations and the species continuity equations associated with a finite rate combustion model. Previous analyses of this concept have been based on inviscid assumptions. The present results indicate that viscous effects are of primary importance; in all the cases studied, shock-induced combustion always started in the boundary layer. The effects of Mach number, mixture composition, pressure, and turbulence are investigated for various configurations. Two types of combustion processes, one stable and the other unstable, have been observed depending on the inflow conditions. In the unstable case, a detonation wave is formed, which propagates upstream and unstarts the ram accelerator. In the stable case, a solution that converges to steady-state is obtained, in which the combustion wave remains stationary with respect to the ram accelerator projectile. The possibility of stabilizing the detonation wave by means of a backward facing step is also investigated. In addition to these studies, two numerical techniques have been tested. These two techniques are vector extrapolation to accelerate convergence, and a diagonal formulation that eliminates the expense of inverting large block matrices that arise in chemically reacting flows.

Pierre, Christophe (ICOMP), Smith, Todd E. (Sverdrup) and Murthy, Durbha V. (University of Toledo): "Localization of Aeroelastic Modes in Mistuned High-Energy Turbines," ICOMP Report No. 91-11, AIAA-91-3379, NASA TM-104445, June 1991, 36 pages.

The effects of blade mistuning on the aeroelastic vibration characteristics of high-energy turbines are investigated, using the first stage of the oxidizer turbopump in the space shuttle main rocket engine as an example. A modal aeroelastic analysis procedure is used in concert with a linearized unsteady aerodynamic theory that accounts for the effects of the blade thickness, camber, and steady loading. Extreme sensitivity of the dynamic characteristics of mistuned rotors is demonstrated. In particular, the aeroelastic modes become localized to a few blades, possibly leading to rogue blade failure, and the locus of the aeroelastic eigenvalues loses its structure when small mistuning (of the order present in actual rotors) is introduced. Perturbation analyses that yield physical insights into these phenomena are presented. A powerful but easily calculated stochastic sensitivity measure that allows the global prediction of mistuning effects is developed.

Pierre, Christophe (ICOMP) and Murthy, Durbha V. (University of Toledo): "Aeroelastic Modal Characteristics of Mistuned Blade Assemblies: Mode Localization and Loss of Eigenstructure," ICOMP Report No. 91-12, NASA TM-104519, July 1991, 38 pages.

An investigation of the effects of small mistuning on the aeroelastic modes of bladed-disk assemblies with aerodynamic coupling between blades is presented. The cornerstone of the approach is the use and development of perturbation methods that exhibit the crucial role of the interblade coupling and yield general findings regarding mistuning effects. It is shown that blade assemblies with weak aerodynamic interblade coupling are highly sensitive to small blade mistuning, and that their dynamics is qualitatively altered in the following ways: the regular pattern that characterizes the root locus of the tuned aeroelastic eigenvalues in the complex plane is totally lost; the aeroelastic mode shapes become severely localized to only a few blades of the assembly and lose their constant interblade phase angle feature; curve veering phenomena take place when the eigenvalues are plotted versus a mistuning parameter.

Liou, William W. (ICOMP) and Morris, P. J. (Pennsylvania State University): "A Comparison of Numerical Methods for the Rayleigh Equation in Unbounded Domains," ICOMP Report No. 91-13, CMOTT-91-4, NASA TM-105179, August 1991, 34 pages.

A second-order finite difference and two spectral methods including a Chebyshev tau and a Chebyshev collocation method have been implemented to determine the linear hydrodynamic stability of an unbounded shear flow. The velocity profile of the basic flow in the stability analysis mimics that of a one-stream free mixing layer. Local and global eigenvalue solution methods are used to determine individual eigenvalues and the eigenvalue spectrum, respectively. The calculated eigenvalue spectrum includes a discrete mode, a continuous spectrum associated with the equation singularity and a continuous spectrum associated with the domain unboundedness. The efficiency and the accuracy of these discretization methods in the prediction of the eigensolutions of the discrete mode have been evaluated by comparison with a conventional shooting procedure. Their capabilities in mapping out the continuous eigenvalue spectra are also discussed.

Jiang, Bo-Nan (ICOMP) and Sonnad, Vijay (IBM Corp.): "Least-Squares Solution of Incompressible Navier-Stokes Equations With the P-Version of Finite Elements," ICOMP Report No. 91-14, NASA TM-105203, September 1991, 14 pages.

A p-version of the least-squares finite element method, based on the velocity-pressure-vorticity formulation, is developed for solving steady-state incompressible viscous flow problems. The resulting system of symmetric and positive definite linear equations can be solved satisfactorily with the conjugate gradient method. In conjunction with the use of rapid operator application which avoids the formation of either element or global matrices, it is possible to achieve a highly compact and efficient solution scheme for the incompressible Navier-Stokes equations. Numerical results are presented for 2D flow over a backward-facing step. The effectiveness of simple outflow boundary conditions is also demonstrated.

Lang, Nancy J. (ICOMP) and Shih, Tsan-Hsing (ICOMP): "A Critical Comparison of Two-Equation Turbulence Models," ICOMP Report No. 91-15, CMOTT-91-05, NASA TM-105237, September 1991, 34 pages.

Several two-equation models have been proposed and tested against benchmark flows by various researchers. For each study, different numerical methods or codes were used to obtain the results which were reported to be an improvement over other models. However, these comparisons may be overshadowed by the different numerical schemes used to obtain the results. With this in mind, several existing two-equation turbulence models, including κ-ε, κ-τ, κ-ω and q-ω models, are implemented into a common flow solver code for near wall turbulent flows. Calculations were carried out for low Reynolds number, 2D, fully developed channel and boundary layer flows. The quality of each model is based on several criterion including robustness and accuracy of predicting the turbulent quantities.

Yang, Zhigang (ICOMP) and Shih, Tsan-Hsing (ICOMP): "A κ-ε Modeling of Near Wall Turbulence," ICOMP Report No. 91-16, CMOTT-91-06, NASA TM-105238, September 1991, 14 pages.

A κ-ε model is proposed for turbulent wall bounded flows. In this model, the turbulent velocity scale and turbulent time scale are used to define the eddy viscosity. The time scale is shown to be bounded from below by the Kolmogorov time scale. The dissipation equation is reformulated using this time scale, removing the need to introduce the pseudodissipation. A damping function is chosen such that the shear stress satisfies the near wall asymptotic behavior. The model constants used are the same as the model constants in the commonly used high turbulent Reynolds number κ-ε model. Thus, when it is far away from the wall, the proposed model reduces to the standard κ-ε model. Fully developed turbulent channel flows and turbulent boundary layer flows over a flat plate at various Reynolds-

numbers are used to validate the model. The model predictions are found to be in good agreement with the DNS data.

Research Briefs - 1990: "Center for Modeling of Turbulence and Transition (CMOTT)," ICOMP Report No. 91-17, CMOTT-91-07, NASA TM-105243, October 1991, 170 pages.

This research brief contains the progress reports of the CMOTT Research staff from May 1990 to May 1991. It is intended to be an informational report of the CMOTT activities as well as an annual report to ICOMP and NASA. The Center for Modeling of Turbulence and Transition (CMOTT), a cooperative turbulence research team, was formally established in May of 1990 as a focal group within the Institute of Computational Mechanics for Propulsion (ICOMP). The location of CMOTT is shown in the organization structure chart in Appendix A. The objectives of the CMOTT are to develop, validate and implement the models for turbulence and boundary-layer transition in the practical engineering flows. The flows of interest are three-dimensional, incompressible and compressible flows with chemistry. The schemes being studied include the two-equation (e.g., κ-ε) and algebraic Reynolds-stress models, the full Reynolds-stress (or second moment closure) models, the probability density function (pdf) models, the Renormalization Group Theory (RNG) and Direct Interaction Approximation (DIA), the Large Eddy Simulation (LES) and Direct Numerical Simulation (DNS).

Arnone, Andrea (ICOMP), Liou, Meng-Sing (NASA Lewis) and Povinelli, L. A. (NASA Lewis): "Multigrid Calculation of 3D Viscous Cascade Flows," ICOMP Report No. 91-18, NASA TM-105257, October 1991, 24 pages.

A 3D code for viscous cascade flow prediction has been developed. The space discretization used a cell-centered scheme with eigenvalue scaling to weigh the artificial dissipation terms. Computational efficiency of a four-stage Runge-Kutta scheme is enhanced by using variable coefficients, implicit residual smoothing, and a full multigrid method. The Baldwin-Lomax eddy-viscosity model is used for turbulence closure. A zonal, non-periodic grid is used to minimize mesh distortion in and downstream of the throat region. Applications are presented for an annular vane with and without end wall contouring, and for a large-scale linear cascade. The calculation is validated by comparing with experiments and by studying grid dependency.

Liou, William W. (ICOMP) and Shih, Tsan-Hsing (ICOMP): "On the Basic Equations for the Second-Order Modeling of Compressible Turbulence," ICOMP Report No. 91-19, CMOTT 91-08, NASA TM-105277, October 1991, 26 pages.

Equations for the mean and the turbulent quantities for compressible turbulent flows are derived in this report. Both the conventional Reynolds average and the mass-weighted, Favre average were employed to decompose the flow variable into a mean and a turbulent quantity. These equations are to be used later in developing second-order Reynolds stress models for high-speed compressible flows. A few recent advances in modeling some of the terms in the equations due to compressibility effects are also summarized.

Demuren, Ayodeji O. (ICOMP): "Multigrid Acceleration and Turbulence Models for Computations of 3D Turbulent Jets in Crossflow," ICOMP Report No. 91-20, NASA TM 105306, CMOTT 91-09, November 1991.

A multigrid method is presented for the calculation of three-dimensional turbulent jets in crossflow. Turbulence closure is achieved with either the standard k-ε model or a Reynolds Stress Model (RSM). Multigrid acceleration enables convergence rates which are far superior to that for a single grid method to be obtained with both turbulence models. With the k-ε model the rate approaches that for laminar flow, but with RSM it is somewhat slower. The increased stiffness of the system of equations in the latter may be responsible. Computed results with both turbulence models are compared with experimental

data for a pair of opposed jets in crossflow. Both models yield reasonable agreement with mean flow velocity but RSM yields better prediction of the Reynolds stresses.

Afolabi, Dare (ICOMP): "Modal Interaction in Linear Dynamic Systems Near Degenerate Modes," ICOMP Report No. 91-21, NASA TM-105315, November 1991, 46 pages.

In various problems in structural dynamics, the eigenvalues of a linear system depend on a characteristic parameter of the system. Under certain conditions, two eigenvalues of the system approach each other as the characteristic parameter is varied, leading to modal interaction. In a system with "conservative coupling", the two eigenvalues eventually repel each other, leading to the curve veering effect. In a system with "non-conservative coupling", the eigenvalues continue to attract each other, eventually colliding, leading to eigenvalue degeneracy. We study modal interaction in linear systems with conservative and non-conservative coupling using singularity theory, sometimes known as catastrophe theory. Our main result is this: eigenvalue degeneracy is a cause of instability; in systems with conservative coupling it induces only geometric instability, whereas in systems with non-conservative coupling eigenvalue degeneracy induces both geometric and elastic instability. Illustrative examples of mechanical systems are given.

Stewart, Mark E. M. (ICOMP): "A 2D Euler Solution for an Unbladed Jet Engine Configuration," ICOMP Report No. 91-22, NASA TM-105329, 1991.

A 2D, nonaxisymmetric Euler solution in a geometry representative of a jet engine configuration without blades is presented. The domain, including internal and external flow, is covered with a multiblock grid. In order to contruct this grid, a domain decomposition technique is used to subdivide the domain, and smooth grids are dimensioned and placed in each block. The Euler solution is verified by examining five theoretical properties. The results demonstrates techniques for performing numerical solutions in complex geometries and provides a foundation for complete engine throughflow calculations.

Stewart, Mark E. M. (ICOMP): "Euler Solutions for an Unbladed Jet Engine Configuration," ICOMP Report No. 91-23, NASA TM-105332, October 1991, 8 pages.

An Euler solution for an axisymmetric jet engine configuration without blade effects is presented. The Euler equations are solved on a multiblock grid which covers a domain including the inlet, bypass duct, core passage, nozzle, and the far field surrounding the engine. The simulation is verified by considering five theoretical properties of the solution. The solution demonstrates both multiblock grid generation techniques and a foundation for a full jet engine throughflow calculation.

Brand, Helmut R. (Universität Essen and Los Alamos National Laboratory) and Deissler, Robert J. (ICOMP): "The Eckhaus and the Benjamin-Feir Instability Near a Weakly Inverted Bifurcation," ICOMP Report No. 91-24, NASA TM-105334, November 1991, 16 pages.

We investigate how the criteria for two prototype instabilities in one-dimensional pattern-forming systems, namely for the Eckhaus instability and for the Benjamin-Feir instability, change as one goes from a continuous bifurcation, to a spatially periodic or spatially and/or time periodic state, to the corresponding weakly inverted, i.e. hysteretic, cases. We also give the generalization to two-dimensional patterns in systems with anisotropy as they arise for example for hydrodynamic instabilities in nematic liquid crystals.

Shih, Tsan-Hsing (ICOMP) and Lumley, John L. (Cornell University): "A Critical Comparison of Second Order Closures with Direct Numerical Simulation of Homogeneous Turbulence," ICOMP Report No. 91-25, NASA TM 105351, CMOTT 91-10, November 1991, 62 pages.

Recently several second order closure models have been proposed for closing the second moment equations, in which the velocity-pressure gradient (and scalar-pressure gradient) tensor and the dissipation rate tensor are the two of the most important terms. In the literature, these correlation

tensors are usually decomposed into a so called rapid term and a return-to-isotropy term. Models of these terms have been used in global flow calculations together with other modeled terms. However, their individual behavior in different flows have not been fully examined because they are unmeasurable in the laboratory. Recently, the development of direct numerical simulation (DNS) of turbulence has given us the opportunity to do this kind of study. With the direct numerical simulation, we may use the solution to exactly calculate the values of these correlation terms and then directly compare them with the values from their modeled formulations (models). In this paper, we make direct comparisons of five representative rapid models and eight return-to-isotropy models using the DNS data of forty five homogeneous flows which were done by Rogers et al. (1986) and Lee et al. (1985). The purpose of these direct comparisons is to explore the performance of these models in different flows and identify the ones which give the best performance. The paper also describes the modeling procedure, model constraints, and the various evaluated models. The detailed results of the direct comparisons are discussed, and a few concluding remarks on turbulence models are given.

Deissler, Robert J. (ICOMP) and Oron, Alexander (Technion-Israel Institute of Technology): "Stable Localized Patterns in Thin Liquid Films," ICOMP Report No. 91-26, NASA TM 105352, November 1991, 14 pages.

We study a 2D nonlinear evolution equation which describes the 3D spatiotemporal behavior of the air-liquid interface of a thin liquid film lying on the underside of a cooled horizontal plate. We show that the Marangoni effect can stabilize the destabilizing effect of gravity (the Rayleigh-Taylor instability) allowing for the existence of stable localized axisymmetric solutions for a wide range of parameter values. Various properties of these structures are discussed.

Jorgenson, Phillp (NASA Lewis) and Turkel, Eli (ICOMP): "Central Difference TVD and TVB Schemes for Time Dependent and Steady State Problems," ICOMP Report No. 91-27, NASA TM 105357, AIAA 92-0053, 1991, 12 pages.

We use central differences to solve the time dependent Euler equations. The schemes are all advanced using a Runge-Kutta formula in time. Near shocks a second difference is added as an artificial viscosity. This reduces the scheme to a first order upwind scheme at shocks. The switch that is used guarantees that the scheme is locally TVD. For steady state problems it is usually advantageous to relax this condition. Then small oscillations do not activate the switches and the convergence to a steady state is improved. To sharpen the shocks different coefficients are needed for different equations and so a matrix valued dissipation is introduced and compared with the scalar viscosity. The connection between this artificial viscosity and flux limiters is shown. Any flux limiter can be used as the basis of a shock detector for an artificial viscosity. We compare the use of the van Leer, van Albada, minmod, superbee and the "average" flux limiters for this central difference scheme. For time dependent problems we need to use a small enough time step so that the CFL was less than one even though the scheme was linearly stable for larger time steps. Using a TVB Runge-Kutta scheme yields minor improvements in the accuracy.

Michelassi, Vittorio (ICOMP) and Shih, Tsan-Hsing (ICOMP): "Elliptic Flow Computation by Low Reynolds Number Two-Equation Turbulence Models," ICOMP Report No. 91-28, NASA TM 105376, CMOTT 91-11, December 1991, 36 pages.

A detailed comparison of ten low-Reynolds-number k-ε models is carried out. The flow solver, based on an implicit approximate factorization method, is designed for incompressible, steady 2D flows. The conservation of mass is enforced by the artificial compressibility approach and the computational domain is discretized using centered finite differences. The turbulence model predictions of the flow past a hill are compared with experiments at $Re = 1.4 \cdot 10^6$. The effects of the grid spacing together with the numerical efficiency of the various formulations are investigated. The results show that the models provide a satisfactory prediction of the flow field in the presence of a favourable pressure gradient, while the accuracy rapidly deteriorates when a strong adverse pressure gradient is encountered. A newly proposed model form that does not explicitly depend on the wall distance seems promising for application to complex geometries.

Jiang, Bo-Nan (ICOMP) and Povinelli, Louis A. (NASA Lewis): "Optimal Least-Squares Finite Element Method for Elliptic Problems," ICOMP Report No. 91-29, NASA TM 105382, December 1991, 18 pages.

In this paper, we propose an optimal least-squares finite element method for 2D and 3D elliptic problems and discuss its advantages over the mixed Galerkin method and the usual least-squares finite element method. In the usual least-squares finite element method, the second-order equation $-\nabla\cdot(\nabla u)+u=f$ is recast as a first-order system $-\nabla\cdot p+u=f$, $\nabla u-p=o$. Our error analysis and numerical experiments show that, in this usual least-squares finite element method, the rate of convergence for flux p is one-order lower than optimal. In order to get an optimal least-squares method, the irrotationality $\nabla\times p=O$ should be included in the first-order system.

SEMINARS

(* = CMOTT Seminars)

Afolabi, Dare (Purdue University - Indianapolis): "Elastic Stability of Rotating Systems and Catastrophe Theory"

Many problems in engineering analysis that seem at first glance to be unrelated are often seen, upon deeper analysis, to have a common basis for their solution. In this presentation, we discuss the fundamental reason why rotating systems are sometimes unstable. Using catastrophe theory, we are able to find a common cause of instability in systems that may appear to be unrelated. For example:

rotating bladed discs, fans, etc.
rotating shafts, shaft-disk systems, etc.
rotating disks, platters, etc.
gyroscopes, gyroscopic systems, spinning tops, etc.

By understanding the fundamental cause of instability in these and similar systems, the structural dynamics engineer will be in a better position to design active or passive means of control more effectively.

Baker, A. J. (University of Tennessee): "On A Non-Dissipative Weak Statement CFD Algorithm for Unsteady Incompressible Navier-Stokes Equations"

The constraint of incompressibility adds a significant mathematical twist to construction of a CFD algorithm. Classical CFD constructions include pseudo-compressibility, pressure relaxation, and penalty methods among others. A weak statement theory permits resolution of details for any pressure correction/relaxation method independent of any specific (spatial) semi-discretization. Thereafter, finite volume and finite element discretizations may be identified, which in concert with a time-integration algorithm produces a computable algebraic statement.

This seminar will highlight the theoretical development, and the subsequent sequence of decisions leading to an intrinsically non-dissipative, tri-linear basis finite element CFD construction exhibiting robust convergence and good accuracy in time accurate unsteady formulation. Numerical results for several benchmark problems, including a buoyancy-dominated 3D problem, verify performance aspects. The theory/code construction should be amenable to subsonic propulsion applications of NASA interest.

Chang, C. L. (Cleveland State University) and Jiang, Bo-nan (University of Texas, Austin): "Least-Squares Finite Element Methods for Fluid Mechanics"

Introduce the least-square finite element methods for 2D incompressible flows and steady-state pure convection problems. Upon introducing an additional variable, $\omega = \text{curl } \underline{u}$, the second order Navier-Stokes equations can be rewritten as a first order system. Then the computational scheme is derived by minimizing the residue of the L_2-norm of equations. The L_1 finite element method captures 2D discontinuities in bands of elements that are only one element wide on both coarse and fine meshes. The solution of the L_1 method has no smearing and no oscillation and is very accurate.

Deane, Anil (Princeton University): "Application of the Karhunen-Loeve Expansion to Some Internal and External Flows: Dynamics and Symmetry"

The Karhunen-Loeve procedure is used to obtain orthogonal basis functions for various flows, including the wakes of a circular cylinder and airfoil, a channel with a cross-stream groove, and thermal convection. These orthogonal functions are used to obtain dynamical equations through a Galerkin projection for these flows. The stability, bifurcation properties and long- and short-term predictive capabilities of these model equations are described. Emphasis in the talk is placed on the flow past a circular cylinder, where the numerical simulations indicate a fast ordered transition through a sequence of period doubled states which possess certain symmetries. The symmetry properties of the associated vector fields and attempts to obtain the entire

sequence of transitions is described as well as the difficulties inherent in the method and their resolution. Extensions to include the study of "coherent structures" and their dynamics in more complex flows are pointed out.

***Deissler, Robert G. (Lewis Research Academy):** "Turbulence and Deterministic Chaos"

Several turbulent and nonturbulent solutions of the Navier-Stokes equations are obtained. The unaveraged equations are used numerically in conjunction with tools and concepts from nonlinear dynamics, including time series, phase portraits, Poincare sections, largest Liapunov exponents, power spectra, and strange attractors.

Initially neighboring solutions for a low Reynolds-number fully developed turbulence are compared. The turbulence, which is fully resolved, is sustained by a nonrandom time-independent external force. The solutions, on the average, separate exponentially with time, having a positive Liapunov exponent. Thus, the turbulence is characterized as chaotic.

In a search for solutions which contrast with the turbulent ones, the Reynolds number (or strength of the forcing) is reduced. Several qualitatively different flows are noted. These are, respectively, fully chaotic, complex periodic, weakly chaotic, simple periodic, and fixed-point. Of these, we classify only the fully chaotic flows as turbulent. those flows have both a positive Liapunov exponent and Poincare sections without pattern. By contrast, the weakly chaotic flows, although having positive Liapunov exponents, have some pattern in their Poincare sections. The fixed-point and periodic flows are nonturbulent, since turbulence, as generally understood, is both time-dependent and aperiodic.

Gajjar, J. S. B. (University of Exeter): "Nonlinear Evolution of Modes in a Compressible Boundary Layer and in the Flow Over Isotorpic and Anisotropic Compliant Walls"

We consider two different problems; the nonlinear evolution of an oblique mode in an insulated or heated/ cooled flat plate boundary layer; and the nonlinear evolution of the travelling-wave flutter modes in the flow over isotropic compliant walls. In both problems, the growth rates are assumed to be small so that the ideas of nonlinear critical layers can be applied. A long wave-length analysis is carried out and it shows that the evolution of the modes is described by the unsteady nonlinear critical layer equations. Explicit expressions for the linear growth rates are derived and compared with numerically calculated values. A new spectral method for the solution of the nonlinear problem will be discussed and some results presented.

Gustavsson, L. Hakan (Lulea University of Technology): "Growth and Control of 3D Disturbances in Channel Flow"

The development of 3D disturbances in channel flow is considered. In particular the forcing of normal vorticity by normal velocity is studied and it is shown that the kinetic energy associated with the normal vorticity can by far exceed the initial energy, even if the participating modes are damped. Characteristics of the growth such as growth areas in wave number space, spanwise selectivity and phase velocities, are first presented together with some non-linear results. Comparison is also made with relevant experiments. Then the possibility of countering the growth by introducing initial normal vorticity is considered and the connection to surface structures is discussed.

Hafez, M. (University of California, Davis): "Some Physical and Numerical Aspects of Compressible and Incompressible Aerodynamic Flow Simulation"

This talk will include a discussion of a new unified approach for compressible and incom-pressible flows using unstructured grids and a finite element least squares formulation. Artificial viscosity necessary for numerical stability and capturing discontinuities are intro-duced explicitly. The associated artificial vorticity is minimized by choosing the form of the artificial viscosity and by a proper treatment of numerical boundary conditions. For steady-state calculations, a convergence acceleration technique based on Wynn's ε-algorithm is demonstrated to be effective for a wide range of Mach and Reynolds numbers.

Hussein, H. J. (Vanderbilt University): "An Experiment To Evaluate the Response of Isotropic Turbulence to Anisotropic Forcing at the Energy Containing Wavenumbers"

Experimental results on the non-linear interscale couplings are reported on the response of isotropic turbulence to anisotropic excitations at specified scales. The experiments were performed in grid turbulence in a low turbulence intensity wind tunnel and the anisotropic forcing was performed with acoustic waves. The spectral tensor components and the streamwise mean-square derivatives are reported for both the case with anisotropic forcing at two eddy turnover times and the case without loading. These results are used to test recent theories on distant triadic interactions which couple large to small scales and predict the persistance of deviations from local isotropy at high Reynolds numbers.

Hussein, H. J. (Vanderbilt University): "Evaluation of Local Isotropy in a Turbulent Plane Jet Using Moving Hot-Wire Measurements"

An experimental investigation of the small scales in a turbulent planar jet using flying hot-wire probes is presented. The overall objective is to provide a basis for evaluating the characteristics of the dissipative scales of turbulence. This involves assessment of local isotropy using the measurements of the spatial derivatives of the velocity field.

Janus, J. Mark (Mississippi State University): "Unsteady Flowfield Simulation of Ducted Prop-Fan Configurations"

NASA has been active in investigating advanced turboprop propulsion systems and have demonstrated the large savings in fuel consumption possible using these advanced turboprop systems compared to conventional turbofan systems. Most recently, interest has shifted toward ducted rather than unducted designs. Presently, there is a computational fluid dynamic (CFD) effort to extend flow analysis software developed for the time-accurate simulation of unducted geometries by incorporating additional domain decomposition mechanisms to enable the simulation of unsteady ducted prop-fan flows (i.e., combined external and internal flow).

The speaker will briefly discuss the numerics of the flow solver which include the equation formulation, numerical flux at cell faces for this cell-centered finite-volume scheme, and the implicit solution algorithm. An additional discussion topic will be the dynamic multiblock grid approach (techniques that have been developed for these particular types of problems involving rotating blocks). Specifically, this will include the extensions made to the tools developed for unducted configurations (i.e., radial partitioning) to enable the modeling of both the external and internal flowfields of complex ducted rotating configurations.

Configurations have been investigated thus far. One consisted of a standard eight-bladed single-rotating prop-fan with SR7 blade design encircled by a short cowl. This configuration was simply a test-case geometry for which no experimental data exists. Another configuration under investigation is that of a single fan stage (rotor and stator) experimentally tested by NASA (NASA 1.15 Pressure Ratio Fan State). This configuration consists of 12 rotor blades and 32 stators enclosed by a nacelle which essentially forms a fan stage (1.15 total pressure ratio) followed by a flow nozzle.

Jeng, San-Mou (University of Tennessee Space Institute): "Spray Combustion Model for Liquid Rocket Engine"

The progress in modelling a bipropellant liquid-rocket combustor will be discussed. The developed CFD code for liquid-rocket-spray combustion was based on the KIVA code, which was originally developed for diesel engine combustion. Numerous state-of-the-art submodels for the N_2H_4/LOX combustion process have been assembled and incorporated. These models include the spray atomization process of a pintle injector, monopropellant and bipropellant droplet combustion, and gas-phase chemistry. A parametric study of N_2H_4/LOX engine performance using different spary injector designs will be presented. Also, continuing efforts to improve code performance will be addressed including the implementation of a computationally efficient gas phase solver and the improvement of a stochastic method for analyzing propellant droplet breakup and dispersion processes.

Jeng, San-Mou (University of Tennessee Space Institute): Propellant Droplet Dynamics in Forced Convective Flows

A computational model based on the Arbitrary-Lagrangian-Eulerian (ALE) numerical algorithm has been developed for flows separated by an immiscible free surface. Surface tension forces and interface exchange coefficients (heat, mass, and momentum) were given special consideration in the modelling effort. In this talk, the considered CFD model will be briefly discussed followed by the applications of the developed CFD code. This CFD code has been used to calculate droplet and bubble oscillation, droplet deformation and breakup, heat and mass transfer of nonspherical droplet, and migration of bubbles in liquids due to thermocapillary force. Also, the on-going experimental work on droplet generator for droplet breakup and droplet-droplet collision will be presented.

***Karniadakis, George (Princeton University):** "Computation of Transitional and Turbulent Flows in Complex Geometries"

General, high-order numerical schemes are formulated appropriate for simulation of transitional and turbulent incompressible flows in complex geometries. In particular, the new schemes are based on mixed explicit/implicit stiffly stable time-stepping methods and explicit treatment of the pressure boundary condition that lead to enhanced stability and arbitrarily high-order time-accuracy dictated entirely by the employed integration rule. Hybrid spectral element methods are then used for the spatial discretization of the variable properties governing equations in three-space dimensions. Special Neumann/viscous sponge type boundary conditions are developed for open flows. Large or subgrid scales in the high Reynolds number regime are modelled through renormalization group (RNG) techniques.

Simulations are then performed to study the transitional and fully turbulent stages of spatially developing flows. Here, we consider the flow over a backward-facing step, and the 3D flow past a circular cylinder. For the first flow, the secondary instability is first investigated and the 3D transitional states are computed through direct simulation (DNS). Transport and subgrid RNG models are then employed to simulate the high Reynolds number flow and heat transfer. Comparisons with experimental data in both regimes are presented. For the second flow, 3D equilibria are accurately resolved via DNS; the series of bifurcations followed is simulated until the cylinder wake becomes turbulent. Our results suggest a successive period doubling in the temporal response of the flow, which eventually leads to a disordered state and renders the wake turbulent.

*** Kollman, W. (University of California, Davis):** "PDF Methods for Turbulent Flames at Subsonic and Supersonic Speeds"

PDF methods offer the possibility of treating rigorously highly nonlinear chemical reactions occurring in turbulent flow fields. This property of PDF methods opened the door to the prediction of turbulent combustion flows without using the assumption of chemical equilibrium. The progress achieved in predicting nonpremixed turbulent flames will be outlined. The description of chemical nonequilibrium in the limit of zero Mach-number requires several variables in addition to a conserved scalar determining the mixture composition on the atomic level. The choice of these additional variables depends on the chemistry of the fuel and oxidizer system considered. Their number is limited by computational constraints. The closure problem arising in one-point PDF equations will be discussed in detail and some new theoretical results on the mixing model will be presented. Selected results for low Mach-number flames will be evaluated.

Turbulent reacting flows at supersonic speeds pose several new challenges. The mixing of fuel and oxidizer becomes a critical issue due to the reduced residence time and the reduced turbulence levels at supersonic speeds. Current attempts for the development of PDF methods for supersonic flow will be discussed.

***Mankbadi, Reda R. (Lewis Research Academy):** "A Critical-Layer Theory for Boundary-Layer Transition"

An asymptotic critical-layer theory is developed to study nonlinear interactions of a triad of instability waves leading to boundary-layer transition. This triad consists of a spatially growing plane fundamental wave and a pair of symmetrical, subharmonic oblique waves. The spatial development of the waves is determined by nonlinear viscous flow in the critical layer. The theory successfully captures not only the linear and parametric resonance stages, but also the later fully interactive regime of the transition process including the saturation

and decay stages. The analysis is fully nonlinear, in that all the nonlinearly generated waves that were not originally present, are accounted for in the analysis. The 3D nonlinear modifications of the mean flow are also considered. The theory applies to both ribbon-induced and naturally occurring transition. The analytically obtained amplitude equations are highly accurate but simple enough to be used for practical transition predictions. Results presented explain experimental observations and reveal novel features of the phenomena.

Matsuuchi, Kazuo (University of Tsukuba): "Necessary Condition for the Generation of Puffs"

It is well known that a peculiar turbulent lump called a "puff" appears in a transitional pipe flow. The puff is stable and has a self-sustained mechanism. Several authors have reported on remarkable characteristics of puffs. However, the condition for the generation of puffs has hardly been studied, because it is very difficult to measure the flow porperties at different axial locations without disturbing the downstream condition. In order to know the generation, experiments were made at a Reynolds number 2160. Two specially designed wires were set upstream to measure the initial growth of the puff, and a third was located far downstream for detection of equilibrium puffs. In a variety of inlet conditions of the pipe, the effect of the strength of inlet disturbances on the generation was investigated. As a result, conditions for the generation and growth of puffs were clarified.

Meneveau, Charles (Johns Hopkins University): "The Wavelet Transform and Its Applications, Part 1: Tutorial and Introduction for Beginners"

This lecture is an introduction to the new and exciting field of Wavelet Transform. Applications include signal processing, image and data compression, numerical methods, CFD and turbulence. Therefore, it has potential value to experimentalists as well as to those engaged in computational and analytical work. In this talk we review some basic properties of the wavelet transform, which is a decomposition of scalar fields into modes that exhibit both localization in wavenumber and physical space. We summarize some basic features of the two types of the wavelet transform: (1) the continuous version and (2) the discrete wavelet transform and series.

The fast wavelet transform (requiring O(NlogN) or sometimes even only O(N) operations) is explained and its relation to multiresolution analysis is illustrated. We review applications in science and engineering, and will demonstrate its use in turbulence research by summarizing our recent results pertaining to the energy cascade, that were obtained by processing direct numerical simulation data. We will discuss the potential of wavelet transform methods in future applications in post-processing of data, and specially in numerical solution of differential equations.

Meneveau, Charles (Johns Hopkins University): "The Wavelet Transform and Its Applications, Part 2: Orthonormal Wavelet Analysis of Turbulence"

We will present a brief review of the wavelet transform. A decomposition of turbulent velocity fields into orthonormal wavelets is performed. We introduce a useful statistical tool, the "dual spectrum" which is as easy to obtain as the traditional Fourier spectrum, but gives more complete information about the spatial distribution of the quantities of interest. From the wavelet transformed Navier-Stokes equations we define dynamical quanities such as the kinetic energy, the transfer of kinetic energy and the flux of kinetic energy through position x and scale r. These quantities are then measured from direct numerical simulations of turbulent flows, and their dual spectra are obtained. Our main findings are that the transfer and flux of kinetic energy greatly fluctuate in physical space and negative values for the flux of energy to smaller scales occur in almost 50 percent of the physical space (backscatter). We conclude that orthonormal wavelet analysis of turbulent signals can be performed rather easily, and it can characterize turbulence dynamics in a more complete and physically more meaningful fashion than Fourier spectra alone.

Messiter, Arthur F. (University of Michigan): "Instability of a Supersonic Vortex Sheet"

According to linear theory, a vortex sheet in supersonic flow is neutrally stable if the Mach number is large enough (Miles 1958). When second-order effects are added, the multiple-scales approach described here

gives an inviscid Burgers equation for the evolution of an assumed initial disturbance. Wave steepening leads to the "kink" modes, with weak shock waves and centered expansions, found by Artola & Majda (1987) analytically and by Pedelty & Woodward (1991) numerically.

It should be added that the results obtained are quite limited in various respects. The effects described give a distortion and eventual decay, rather than growth, of the initial disturbance; interaction of modes, not considered here, may lead to further growth (Artola & Majda 1989). But a vortex sheet evidently does not represent the proper long-wave limit for a thin shear layer, which is unstable to weakly nonlinear disturbances that grow in the critical layer (Balsa 1990, 1991). It appears that nonlinearity in the external flow would enter only at a later stage of this shear-layer instability.

***Moin, Parviz (Stanford University):** "A Dynamic Model for Large Eddy Simulation of Turbulent Flows"

One major drawback of the subgrid scale models currently used in large eddy simulations is their inability to correctly represent with a single universal constant different turbulent flows in rotating or sheared flows, near solid walls or in transitional regimes. A new subgrid scale model will be presented which alleviates many of these drawbacks. The model coefficient is inputted dynamically rather than inputted a *priori*. The basic idea in the derivation of the model is the utilization of the spectral information which is computed directly. This rather rich spectral information is not available in methods based on Reynolds-averaged equations. The subgrid scale stresses obtained using the proposed model vanish in laminar flow and at a solid boundary. The results of large eddy simulation of transitional and turbulent channel flow that use the proposed model are in good agreement with the direct numerical simulation data. The same model was applied to the decay of isotropic turbulence with excellent agreement with the experimental data. The model has been extended to compressible flows. A dynamic subgrid scale turbulent Prandtl number was derived. Its dependence on molecular Prandtl number, direction of scalar gradient, and the distance from the wall are in accordance with direct simulation.

Oron, Alex (Technion, Israel Institute of Technology): "Two-Dimensional Waves in Liquid Film Flowing on a Vertical Cylinder"

An amplitude equation is derived, which describes the evolution of a disturbed film interface flowing down an infinite vertical cylinder column. Using a new approach, which accounts for fast spatial changes, the nonlinear evolution of the interface is shown to be governed by an equation associated with the Kuramoto-Sivashinsky equation. It is shown numerically that in the 1D case, for some linearly unstable equilibria the evolving waves break in a finite time.

We next investigate in a laboratory frame the evolution of waves on the surface of a cylinder in the context of a 2D Kuramoto-Sivashinsky equation. The conditions for convective instability of the system are derived. We show that spiral waves emerge when the flow is driven at the inlet with a combination of spiral driving and random noise. In the case of a purely noise-driven flow, spiral-like structures appear occasionally while the main pattern is a straight wave. In most cases the flow tends to be irregular.

Petropoulos, Peter G. (Northwestern University): "Radiation Boundary Conditions for the Simulation of Waves in Layered Media"

The well-known radiation boundary conditions for the numerical simulation of waves are not applicable to problems of wave propagation and scattering in layered media. New radiation boundary conditions of the global type will be presented for the Helmholtz equation in a layered medium, and their implementation in a finite element interior scheme will be discussed.

Pletcher, Richard H. (Iowa State University): "Recent Results in the Numerical Simulation of Unsteady Viscous Flows"

Some recent results in the simulation of unsteady viscous flows will be presented. One of the long-term objectives of this study has been to develop a computational strategy that is efficient and accurate over a range of Mach numbers from the incompressible limit up through transonic speeds. Key elements in achieving compressible formulations that converge in a manner independent of Mach number will be discussed. The

addition of relatively simple pseudo-time terms to the governing equations permit time-accurate solutions to be obtained efficiently at vanishingly small Mach numbers. This computational strategy is being applied in several application areas including the computation of internal viscous flows on unstructured grids, the simulation of three-dimensional liquid sloshing in an orbiting spherical tank, and the numerical simulation of flows in combustion systems. Results from some of these studies will be presented as time permits.

***Speziale, Charles G. (ICASE, NASA Langley):** "A Critical Assessment of Turbulence Modeling"

A variety of turbulence models including zero, one and two-equation models as well as second-order closures will be reviewed. Based on comparisons with results from physical and numerical experiments on homogeneous turbulence, a strong case will be made for the superior predictive capabilities of second-order closure models. It will be shown how some significant improvements in second order closure models have been recently achieved by means of invariance arguments coupled with a dynamical systems approach. Several applications will be discussed including recent extensions to high speed compressible flows that were developed in connection with the National Aerospace Plane Project.

Steger, Joseph L. (University of California, Davis): "Chimera Method for Three-Dimensional Computational Flow Simulation"

The chimera overset grid domain decomposition method has been successfully applied to the simulation of viscous flow about various complex configurations. Although unstructured grid methods of numerical simulation are receiving considerable attention in fluid dynamics, the chimera scheme is providing to be as versatile and can be argued to be more general. Moreover, the composite of overset structured grids that the chimera scheme uses to resolve geometry, flow features, or permit more efficient flow solvers also enhances the use of vectorized and parallel computers. A brief overview of the chimera scheme is given followed by discussion of enhancements now underway.

Tryggvason, Gretar (University of Michigan): "Full Numerical Simulations of Multi-Fluid Flows"

To fully understand the behavior of a multi-fluid system one must have a good insight into the basic micromechanisms that govern the evolution of a single structure (e.g., a bubble or a drop) and the interactions of a few such structures. Full numerical simulations are, in principle, ideally suited to provide this information. Not only are all the quantitative data readily available, but various physical processes can be turned on and off at will. In practice, however, simulations of multi-fluid problems are one of the difficult areas of computational fluid dynamics, and almost all current studies of multi-fluid problems make a number of simplifications. Although simplified models capture some of the important behavior, many of the fundamental processes in multi-fluid flow are fully 3D, and both inertia and viscous effects must be accounted for.

We have recently developed a method for multi-fluid, incompressible flows that appears to be both accurate and robust. The method is based on rather standard finite difference discretization of the flow, but the interface between the two fluids is explicitly tracked by a separate moving, unstructured interface grid. This keeps the interface sharp at all times (no numerical diffusion), and allows the inclusion of surface tension in a natural way. The method has been implemented for 2D as well as fully 3D situations.

Here we describe the method and discuss results of simulations of collisions of two bubbles in a periodic bubble array, the Rayleigh-Taylor instability, and preliminary results on the rise of contaminated bubbles.

Usab, William J. (Purdue University): "A Solution Adaptive Unstructured Scheme for Quasi-3D Flows Through Advanced Turbomachinery Cascades"

This talk will present an overview of research directed toward development of a solution adaptive scheme for the accurate prediction of inviscid quasi-3D flows in advanced compressor and turbine designs. The approach used here combines an explicit finite-volume time-marching scheme for unstructured triangular meshes and an advancing front mesh generation scheme, with a remeshing procedure for adapting the mesh as the solution evolves. Several inviscid 2D and quasi-3D flow problems have been solved using the present solution adaptive formulation. Mesh adapted quasi-3D Euler solutions for three spanwise stations of the NASA rotor 67 transonic fan will be presented.

WORKSHOP ON ENGINEERING TURBULENCE MODELING

Under the auspices of the Center for Modeling and Transition (CMOTT), a day-and-a-half workshop on engineering turbulence modeling was held in August 1991. The stated objectives and outline of the workshop follow:

OBJECTIVES

The purpose of the workshop is to discuss the present status and the future direction of various levels of engineering turbulence modeling related to CFD computations for propulsion. For each level of complication, there are a few turbulence models which represent the state-of-the-art for that level; however, it is important to know their capabilities as well as their deficiencies in CFD computations in order to help engineers select and implement the appropriate models in their real world engineering calculations. This will also help turbulence modelers perceive the future directions for improving turbulence models.

The focus of the workshop will be one-point closure models (i.e. from algebraic models to higher order moment closure schemes and pdf methods) which can be applied to CFD computations. However, other schemes helpful in developing one-point closure models such as RNG, DIA, LES and DNS, will be also discussed to some extent.

OUTLINE OF THE WORKSHOP:

August 21, 1991 (Wednesday)

Registration
Welcome by L. Povinelli

Session I: Turbulence Modeling in CFD and Algebraic Closure Models
Chairman: E. Reshotko

B. E. Launder, "The Current Status of Turbulence Modeling in CFD and its Future Prospects".

D. M. Bushnell, "Comment Paper".

Discussion

D. Wilcox, "The Present State and the Future Direction of Eddy Viscosity Models".

P. Spalart, "Comment Paper".

T. Coakley, "Comment Paper".

Discussion

D. Taulbee, "The Present State and Future Direction of Algebraic Reynolds Stress Models".

A. O. Demuren, "Comment Paper".

Discussion

Lunch Break

Session II: Second Order Closure and PDF Method
Chairman: J. L. Lumley

T.-H. Shih, "The Present State and the Future Direction of Second Order Closure Models for Incompressible Flows".

J. R. Ristorcelli, Jr., "Comment Paper".

C. G. Speziale, "Comment Paper".

Discussion

T. B. Gatski, "The Present State and the Future Direction of Second Order Closure Models for Compressible Flows".

J. Viegas, "Comment Paper".

G. Huang, "Comment Paper".

Discussion

S. Pope, "The Present State and the Future Direction of PDF Methods".

E. E. O'Brien, "Comment Paper".

J. Y. Chen, "Comment Paper".

Discussion

Session III: Unconventional Turbulence Modeling
Chairman: J. H. Ferziger

A. Yoshizawa, "The Present State of DIA Models and Their Impact on One Point Closures".

J. Weinstock, "The Present State of Two-Point Closure Schemes and Their Impact on One Point Closures".

S. Orszag, "The Present State of RNG and its Impact on One Point Closure".

R. R. Mankbadi, "The Present State of Application of RDT to Unsteady Turbulent Flows".

W. K. George, and J. H. Ferziger, "The Role of Experiments and DNS & LES in Supporting Turbulence Modeling Efforts".

Discussion

Concluding Remarks

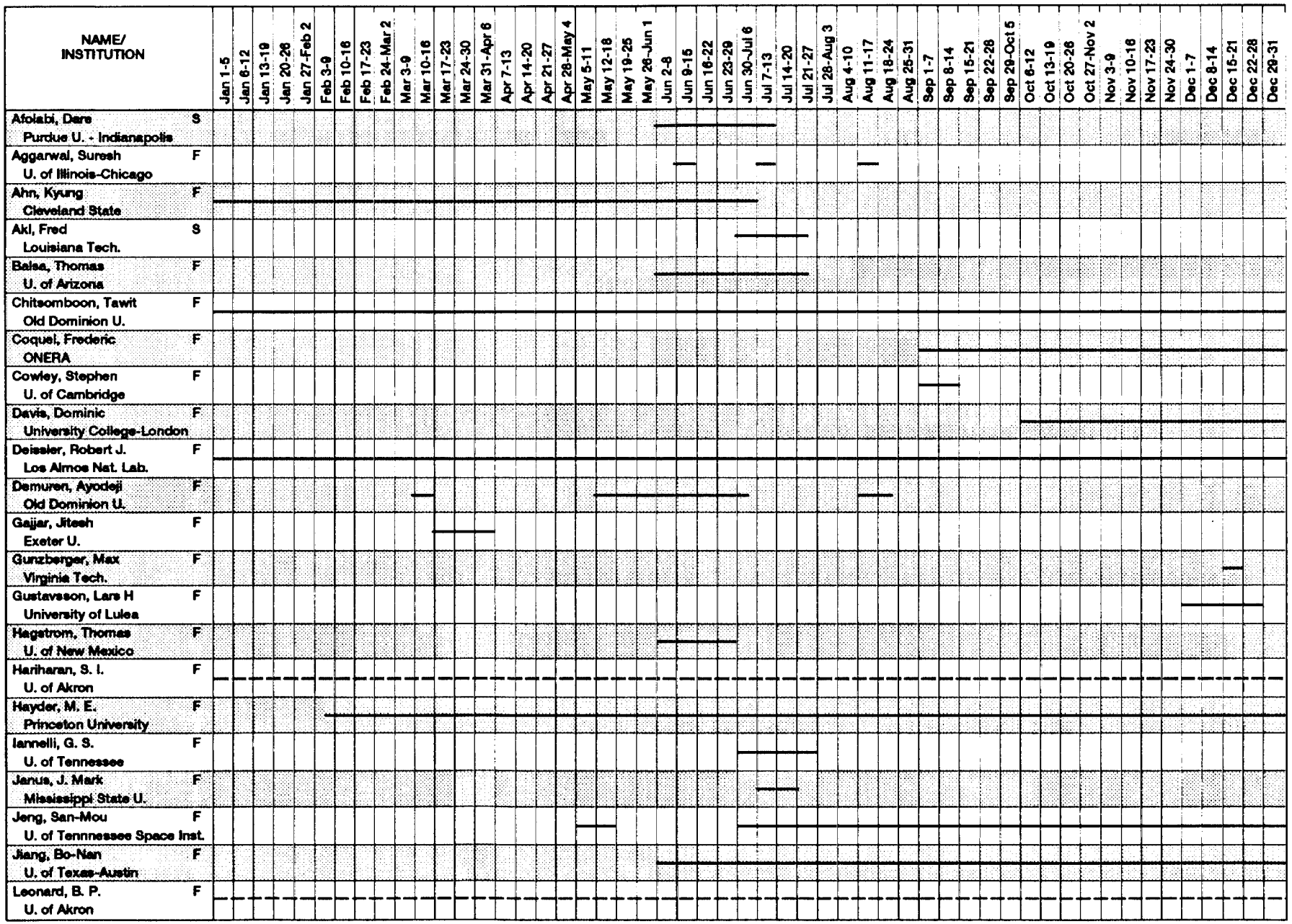

Figure 1.—ICOMP visiting researchers, 1991.

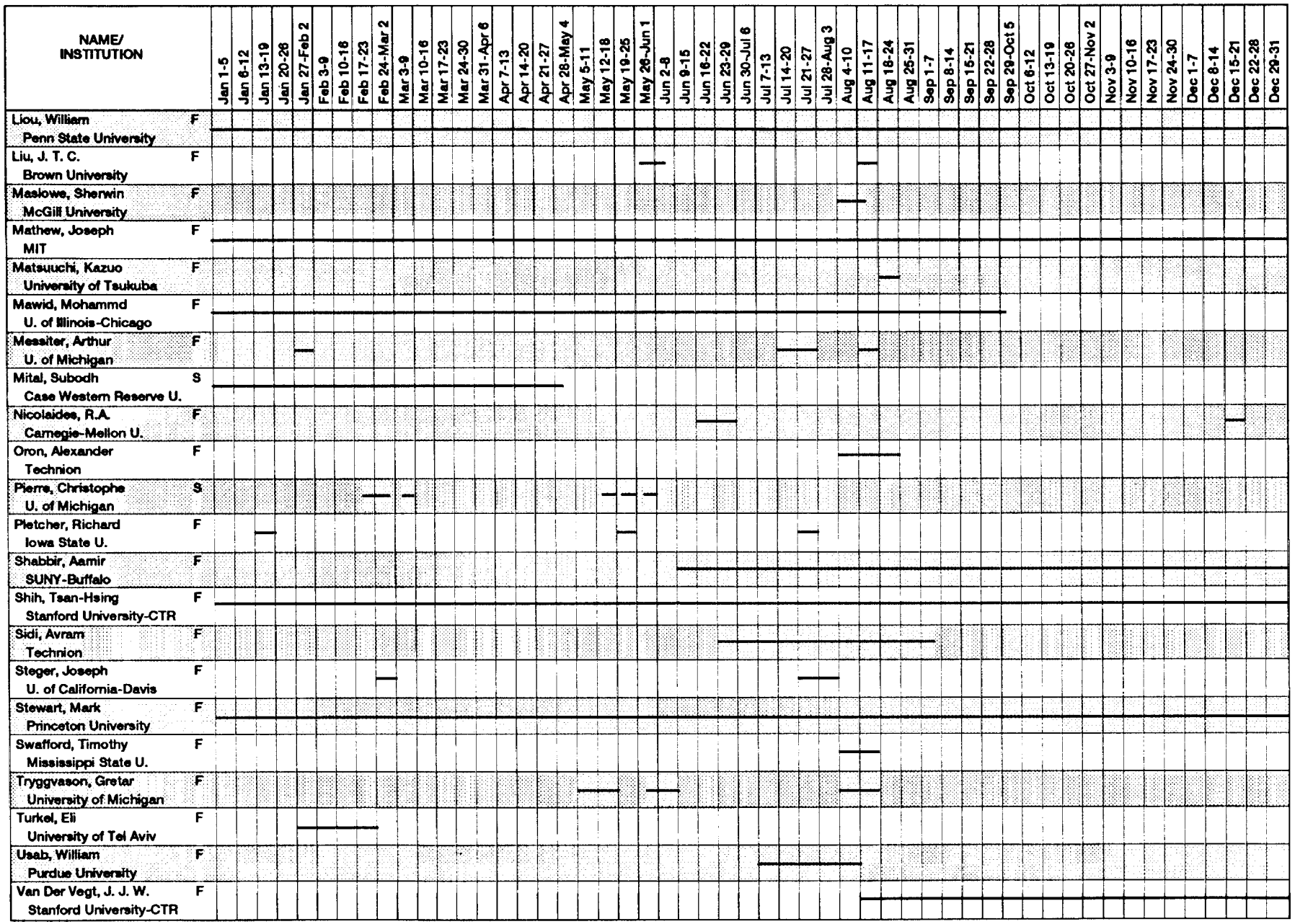

Figure 1.—Continued.

Figure 1.—Concluded.

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

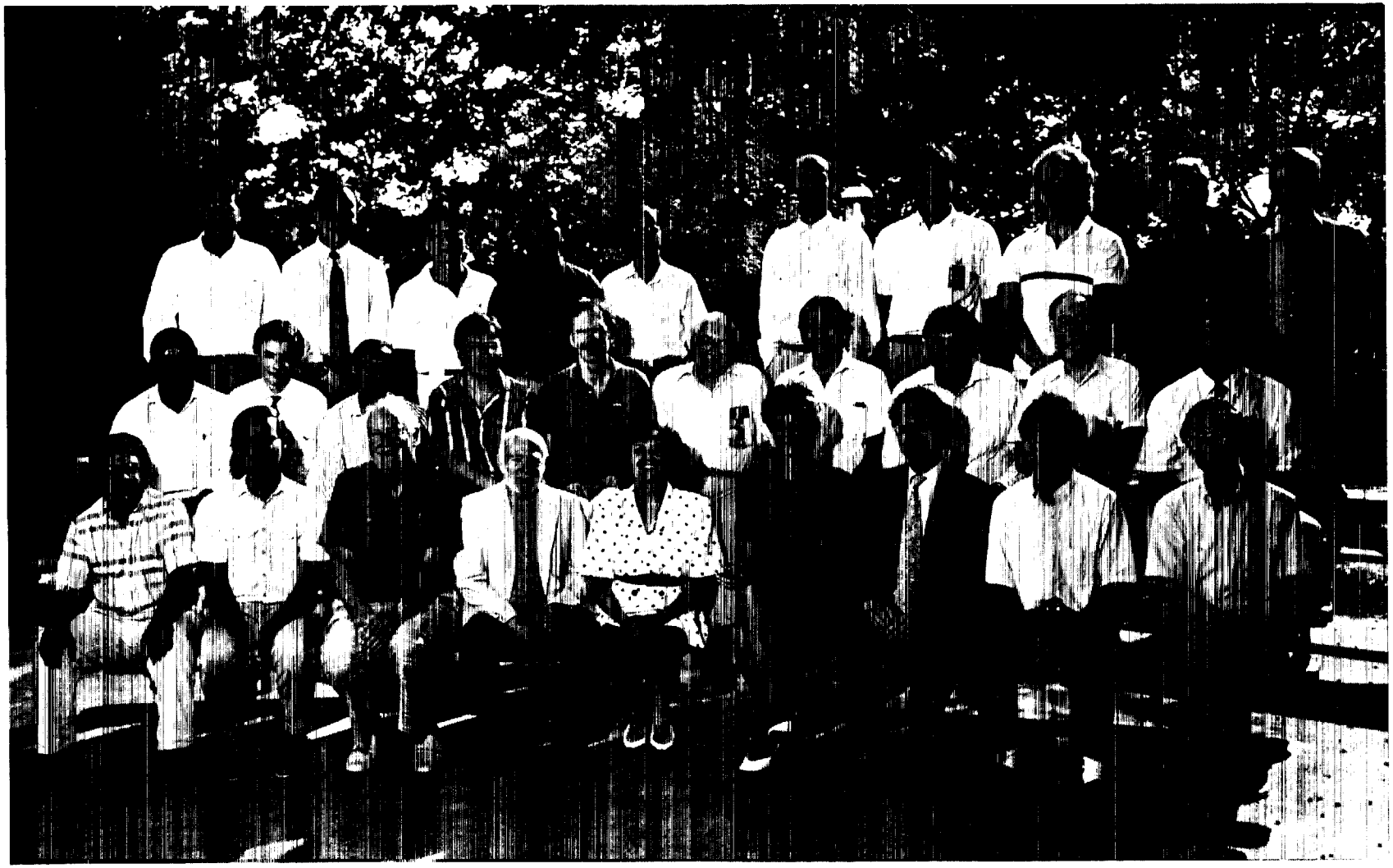

Figure 2.—ICOMP Steering Committee and visiting researchers in July, 1991. First row: S. I. Hariharan; Mark Janus; Charles Feiler; Isaac Greber; Glenda Gamble; Karen Oring; Michael Salkind; Kyung Ahn; T.-H/ Shih. Second row: Dare Afolabi; Joe Iannelli; M.E. Hayder; Shaye Yungster; Robert J. Deissler; Eli Reshotko; Louis Povinelli; Robert Mullen; Avram Sidi; Fred Akl. Third row: Suresh Aggarwal; William Usab; William Liou; Joseph Mathew; Bo-Nan Jiang; Aamir Shabbir; San-Mou Jeng; Greg Swartwout; Thomas Balsa; Mark Stewart.

	UNIVERSITY OR INSTITUTION	NUMBER
1	AKRON	2
2	ARIZONA	1
3	BROWN	1
4	CALIFORNIA, DAVIS	1
5	CAMBRIDGE	1
6	CARNEGIE-MELLON	1
7	CASE WESTERN RESERVE	1
8	CLEVELAND STATE	1
9	CORNELL	1
10	EXETER	1
11	ILLINOIS, CHICAGO	2
12	IOWA STATE	1
13	LOS ALAMOS NAT. LAB	1
14	LOUISIANA TECH	1
15	LULEA	1
16	McGILL	1
17	M.I.T.	1
18	MICHIGAN	4
19	MISSISSIPPI STATE	3
20	NEW MEXICO	1
21	OLD DOMINION	2
22	ONERA	1
23	PENN STATE	1
24	PRINCETON	2
25	PURDUE	1
26	PURDUE, INDIANAPOLIS	1
27	STANFORD (CTR)	2
28	SUNY, BUFFALO	1
29	TECHNION - Israel Institute of Tech.	2
30	TEL AVIV	1
31	TENNESSEE	1
32	TENNESSEE SPACE INST.	1
33	TEXAS, AUSTIN	1
34	TOKYO	1
35	TSUKUBA	1
36	UNIVERSITY COLLEGE, LONDON	1
37	VIRGINIA TECH	1
38	WASHINGTON	1
		49

Figure 3.—Composition of 1991 ICOMP staff-organizations represented.

	1986	1987	1988	1989	1990	1991
PEOPLE	23	43	50	46	47	49
SEMINARS	10	27	39	30	37	26
REPORTS	2	9	22	32	25	29
WORKSHOPS/LECTURE SERIES	1	0	2	1	1	1
PRESENTATIONS	7	0	21	14	15	21
NIVERSITIES OR ORGANIZATIONS	20	28	43	35	36	38

Figure 4.—ICOMP statistics (1986 to 1991).

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188), Washington, DC 20503.

1. AGENCY USE ONLY (*Leave blank*)	2. REPORT DATE	3. REPORT TYPE AND DATES COVERED
		Technical Memorandum

4. TITLE AND SUBTITLE

Institute for Computational Mechanics in Propulsion (ICOMP)
Sixth Annual Report – 1991

5. FUNDING NUMBERS

WU–505–62–21

6. AUTHOR(S)

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)

National Aeronautics and Space Administration
Lewis Research Center
Cleveland, Ohio 44135–3191

8. PERFORMING ORGANIZATION REPORT NUMBER

E–6950

9. SPONSORING/MONITORING AGENCY NAMES(S) AND ADDRESS(ES)

National Aeronautics and Space Administration
Washington, D.C. 20546–0001

10. SPONSORING/MONITORING AGENCY REPORT NUMBER

NASA TM–105612
ICOMP–92–01

11. SUPPLEMENTARY NOTES

Report compiled and edited by Charles E. Feiler, ICOMP Executive Officer, and approved by Louis A. Povinelli, ICOMP Director, NASA Lewis Research Center (work funded under NASA Cooperative Agreement NCC3–233). ICOMP Program Director, Louis A. Povinelli, (216) 433–5818.

12a. DISTRIBUTION/AVAILABILITY STATEMENT

Unclassified - Unlimited
Subject Category 64

12b. DISTRIBUTION CODE

13. ABSTRACT (*Maximum 200 words*)

The Institute for Computational Mechanics in Propulsion (ICOMP) is a combined activity of Case Western Reserve University, Ohio Aerospace Institute (OAI) and the NASA Lewis Research Center in Cleveland, Ohio. The purpose of ICOMP is to develop techniques to improve problem-solving capabilities in all aspects of computational mechanics related to propulsion. This report describes the activities at ICOMP during 1991.

14. SUBJECT TERMS

Numerical analysis; Computer science; Mathematics; Fluid mechanics

15. NUMBER OF PAGES
48

16. PRICE CODE
A03

17. SECURITY CLASSIFICATION OF REPORT	18. SECURITY CLASSIFICATION OF THIS PAGE	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT
Unclassified	Unclassified	Unclassified	